AN-SOF v2.7

Electromagnetic Simulation Tool

© 2000-2012 Golden Engineering

USER'S GUIDE

AN-SOF v2.7

Electromagnetic Simulation Tool

USER'S GUIDE

© 2000-2012 Golden Engineering

AN-SOF® is a registered trademark.

© 2000-2012 Golden Engineering.
All rights reserved. Published in June 2012. Printed in Argentina.

Preface

Congratulations for choosing **AN-SOF®**, the easiest-to-use electromagnetic simulator for the modeling and design of antennas and general wire structures.

AN-SOF® is an innovative software tool for the modeling and simulation of antenna systems and general radiating structures. Transmitting and receiving antennas can be designed and several antenna parameters can be obtained as a function of frequency: input impedance, standing wave ratio (SWR), efficiency, radiated and consumed powers, gain, directivity, beamwidth, front to back ratio, radar cross section (RCS), polarized field components, etc.
The radiation and scattering properties of a structure can be represented in fully angle-resolved 3D patterns. Colored mesh and surface for the clear visualization of radiation lobes are available as well as the traditional polar graphs. Other remarkable features include near-fields in 2D and 3D colored plots, current distributions, reflection coefficients in Smith charts, tapered and insulated wires, large and short antennas over real ground and transmission line modeling. Simulations of curved wire antennas, like helices, spirals and loops can be efficiently performed by means of the Conformal Method of Moments (CMoM), which has been exclusively implemented in AN-SOF®.

To stay informed about new releases and advances in electromagnetic simulation tools, please visit our site at www.antennasoftware.com.ar.

User's Guide

The present *User's Guide* describes **AN-SOF® v2.7** and its many functions in detail. The guide is organized on the stages of electromagnetic simulation, and explains all aspects of using **AN-SOF®** in detail.

On-Line Help

AN-SOF® offers a full help file system to support your use of the product*. Choose **Help/Contents** to display the help file that explains the AN-SOF® program in detail, or choose **Help/Index** to display the help file where you can type the word you are looking for. Both are standard Windows® help files, offering a table of contents and index.

In addition, you can display context-sensitive help by pressing F1 from any command or window.

*Microsoft has released Windows Vista and 7 without WinHelp. Then, context-sensitive help is not available in Windows Vista/7 unless the WinHelp is installed, which is available for download at the Microsoft site
http://www.microsoft.com/download/en/details.aspx?DisplayLang=en&id=5143

Adobe PDF Files

The present *User's Guide* is provided in the AN-SOF® v2.7 installation files as an Adobe PDF file and is accessible from the AN-SOF® program folder on the Windows® Start menu. To open PDF files, you will need Adobe's free Acrobat Reader program, available for download at www.adobe.com.

Disclaimer of Warranty

The technical descriptions, procedures and software included in this *User's Guide* have been developed with the greatest care. They are provided without warranty of any kind. Golden Engineering Ltd. makes no warranties, expressed or implied, that the equations, programs and procedures in this guide or its associated software are free of error, consistent with any particular standard of merchantability, or will meet your requirements for any particular application. They should not be relied on for solving a problem whose incorrect solution could result in injury to a person or loss of property. Any use of the programs or procedures in such a manner is at the user's own risk. Golden Engineering Ltd. disclaims all liability for direct, incidental, or consequential damages resulting from use of the programs or procedures in this guide or the associated software.

Contents

Summary

AN-SOF® is a comprehensive software tool for the analysis and design of antenna systems and general radiating structures.

AN-SOF® calculates the electric currents flowing on metallic wires by means of an improved version of the so-called Method of Moments (MoM). In this method, metallic structures like antennas are described by a set of wires and wire grids. Then, the wires are decomposed into small pieces that are short compared to the wavelength: the segments. An individual segment has usually the form of a short cylindrical wire that approaches the electromagnetic behavior of an electric dipole. Thus, any antenna or metallic structure can be thought of as made of short electric dipoles. When a source is placed at some position on the structure, a current is forced to flow over the wires. This induced current distribution is the first quantity calculated by AN-SOF® in any simulation. Afterwards, the radiated field can be computed as well as the input impedance at the position of the source.

The situation described above is the most common one that can be encountered in the simulation of a transmitting antenna. However, there are several more possibilities that can be handled with AN-SOF®, such as transmitting antennas with multiple voltage and current sources, receiving antennas illuminated by incoming waves, complex antenna arrays, antennas with loading impedances, wires coated with an insulation material, scattered waves by arbitrarily shaped obstacles, ground waves traveling over the soil surface, and virtually any scenario where electromagnetic waves are interacting with metallic objects.

In the case of antennas, several parameters can be obtained as a function of frequency, such as input impedance, standing wave ratio (SWR), efficiency, radiated and consumed powers, gain, directivity, beamwidth, front to back ratio, radar cross section (RCS), linearly and circularly polarized fields, etc.

The geometry of the wire structure can be easily drawn on the screen using dialog boxes for the input data. All wires are placed in 3D space where several 3D-tools with mouse support have been implemented, including zoom, motion and rotation features.

Lumped impedances representing resistors, inductors and capacitors can be placed at arbitrary locations on the structure. Voltage and current generators can be used as sources in the transmitting case, while an incident plane wave of arbitrary incoming direction and polarization can be defined to illuminate an object in the receiving case.

The software provides a suite of dedicated graphical tools that allow for the representation of the results in 2D and 3D plots. The electric currents flowing on a structure can be visualized directly on the wires as a colored intensity map. The radiation pattern in the far-field zone can be displayed either as a

rectangular plot, as a traditional polar plot or as a fully angle-resolved 3D pattern. The radiation lobes in a 3D plot are shown as smooth surfaces with a colored scale that can be superimposed to the antenna geometry for a better interpretation of its directional properties. Near-fields in the proximity of a structure can also be represented with color maps for the electric and magnetic field intensities. Input impedances, admittances, SWR and reflection coefficients can be plotted as a function of frequency in a Smith chart representation.

The AN-SOF® capabilities are not only limited to bare metallic structures, but wires coated with a general material having dielectric and/or magnetic properties can also be simulated. Besides, the skin effect is taken into account when the metallic materials have a non-zero resistivity. Different resistivities at different locations on the structure can be defined.

In the case of curved antennas like loops, helices and spirals, the wire segments composing the structure have a curvature that follows the exact shape of the antenna geometry. Usually a curved antenna is roughly approximated by a broken line with straight segments, thus introducing an input error to the simulation that can never be fixed. Instead of straight wire segments, conformal segments are used in AN-SOF® to exactly follow the contour of curved antennas. This innovation has been coined as the Conformal Method of Moments (CMoM). AN-SOF® is the only electromagnetic simulator that implements the CMoM.

1. Introduction

1.1 Program Description

AN-SOF® is a comprehensive software tool for the modeling and simulation of antenna systems and general radiating structures.

AN-SOF® is intended for solving problems in the following areas:

- Antenna analysis and design.
- Electromagnetic Compatibility (EMC) applications.
- Multiconductor transmission lines.
- Passive circuits and general non-radiating networks.

The program is based on an improved version of the so-called Method of Moments (MoM) for wire structures. Metallic objects like antennas can be modeled by a set of conductive wires and wire grids, as it is illustrated in Fig. 1.1. In the MoM formulation, the wires composing the structure are divided into segments that must be short compared to the wavelength. If a source is placed at a given location on the structure, an electric current will be forced to flow on the segments. The induced current on each individual segment is the first quantity calculated by AN-SOF®.

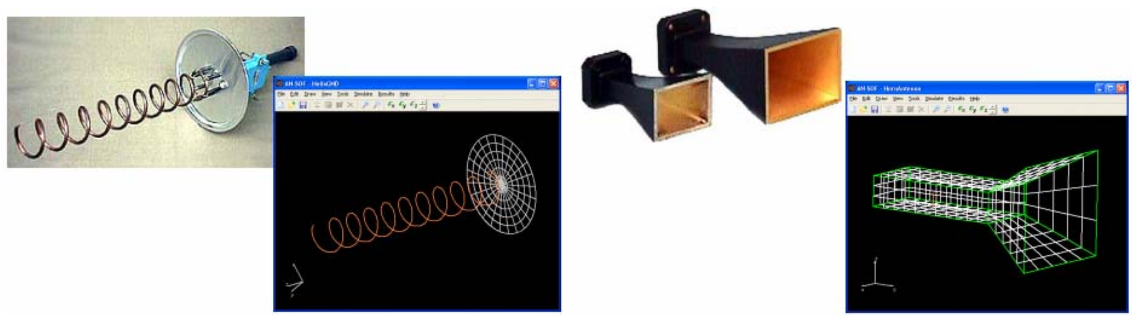


Fig. 1.1: Antennas modeled by means of wires and wire grids.

Once the current distribution has been obtained, the radiated electromagnetic field can be computed in the far- and near-field zones. Input parameters at the position of the source or generator can also be obtained, such as the input impedance, input power, standing wave ratio (SWR), reflection coefficient, transmission loss, etc.

The modeling of the structure can be performed by means of the AN-SOF® specific 3D CAD interface. Electromagnetic fields, currents, voltages, input impedances, consumed and radiated powers, gain, directivity, and several more parameters can be computed in a frequency sweep and plotted in 2D and 3D graphical representations.

In the case of curved antennas like loops, helices and spirals, the MoM method has been improved to account for the exact curvature of wires. In traditional calculations, curved antennas are modeled using straight-line segments with a lot of discontinous wire junctions. This linear approximation to the geometry can be very inefficient in terms of computer memory and the number of calculations to be performed, since several straight segments must be used to reproduce the curvature of smooth curved wires. To overcome this inaccuracy, curved segments that exactly follow the contour of curved antennas are used in AN-SOF®. This innovative technique has been coined as the Conformal Method of Moments (CMoM).

As an example, Fig. 1.2 shows the different approaches to a circular disc obtained by means of the MoM and CMoM methods. Both methods are available in AN-SOF® since the MoM is considered to be a special case of the more general CMoM.

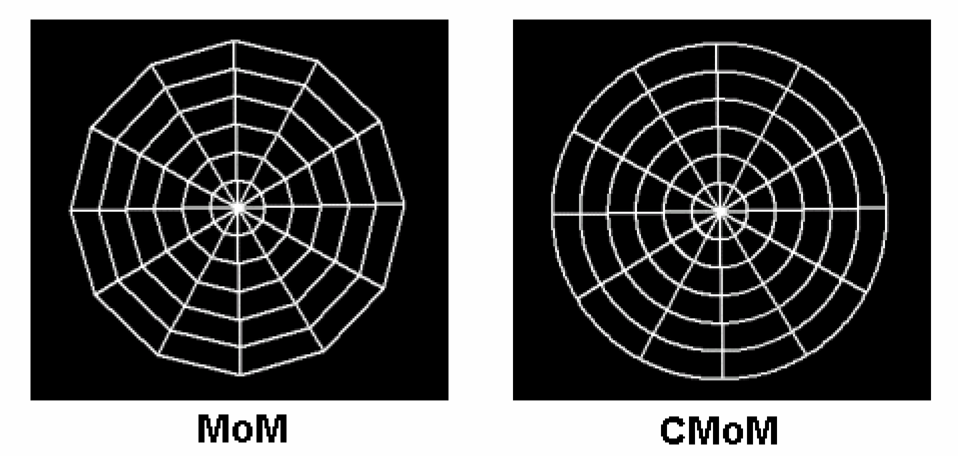


Fig. 1.1: Modeling of a disc by means of the MoM and CMoM methods.

In addition to the CMoM capabilities, advanced mathematical techniques has been implemented in the calculation engine making possible simulations from extremely low frequencies (e.g. electric circuits at 50-60 Hz) to very high ones (e.g. microwave antennas above 1 GHz).

In what follows, a summary of the modeling options and the simulation results that can be obtained from AN-SOF® is presented.

Modeling of metallic structures

1. Metallic structures can be modeled by combining different types of wires and wire grids:

   Wires
   - Straight wire
   - Circular arc
   - Circular loop
   - Helix
   - Quadratic wire
   - Archimedean spiral
   - Logarithmic spiral

   Wire grids
   - Plate
   - Disc
   - Flat ring
   - Cone
   - Truncated cone
   - Cylinder
   - Sphere
   - Paraboloid

2. All types of curved wires can be modeled by means of arced and quadratic segments.
3. Wire grids can be defined using either curved or straight wire segments. Curved segments follow the exact curvature of discs, rings, cones, cylinders, spheres, and parabolic surfaces. Wire grids can be used to model grids and approximate conductive surfaces.
4. Tapered wires with stepped radii can be defined.
5. All wires can be loaded or excited at any position.
6. The structure can also have finite non-zero resistivities (skin effect).

7. Electrical connections of different wires and connections of several wires at one point are possible.
8. Metallic wires in either dielectric or magnetic media can be analyzed.
9. Wires with insulation can be modeled. Dielectric and magnetic coatings are available.
10. The structures can be placed in free space as well as over a perfectly conducting ground plane. The effect of a real ground on the near and far fields radiated from the structure can also be computed.
11. The wire cross-section can either be Circular, Square, Flat, Elliptical or Rectangular.
12. The geometry modeling can be performed in suitable unit systems (um, cm, mm, m, in, ft). Different unit systems can also be chosen for inductance (pH, nH, uH, mH, H) and capacitance (pF, nF, uF, mF, F).

Excitation methods

1. An arbitrary number of voltage sources can be placed at any position, with equal or different amplitudes (rms values) and phases.
2. Current sources (e.g. representing impressed currents) can also be arranged at any positions.
3. The voltage and current sources can have internal impedances.
4. An incident plane wave of arbitrary polarization (linear, circular or elliptical) and direction of incidence can also be used as the excitation.
5. Hertzian electric and magnetic dipoles can also be modeled and used as the excitation.

Frequency options

1. The computation can either be performed for a single frequency, for frequencies taken from a list or for a frequency sweep.
2. The list of frequencies can either be created inside the program or loaded from a text file. It can also be saved to a txt file.
3. Linear and logarithmic frequency sweeps are possible.
4. A suitable unit system can be selected (Hz, KHz, MHz, GHz).

Data input

1. 3D CAD tools are implemented for drawing the structure geometry. Wires, wire grids, discrete generators and lumped loads can easily be added, modified or deleted.
2. The segmentation of the wire geometry is done automatically, but can also be set manually by the user.
3. Any wire can be selected and highlighted by clicking with the right/left mouse button on the screen.
4. Clicking on a wire shows a pop-up menu with several options.
5. Wire connections can easily be performed by means of a *copy/paste* function for the end points of the wires.
6. The source, load element and ground point positions are shown with special 3D-symbols.
7. All dialog boxes check for valid inputs.
8. Rotation, move and zoom functions with mouse support are implemented.
9. The powerful MATLAB® Component Runtime (MCR®) is integrated into the AN-SOF® architecture for getting the fastest calculation speed and, at the same time, the most accurate results.

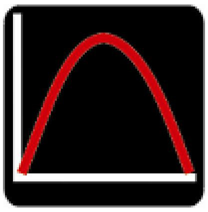
Data output

1. All computed data is written to storage files for a subsequent graphical evaluation.
2. Input impedances, currents, voltages over loads, VSWR, return and transmission losses, radiated and consumed powers, directivity, gain and other system responses are shown as lists in text format and can be plotted vs. frequency. A Smith chart is available for representing impedances and admittances as well as for showing the reflection coefficient and VSWR at the mouse selected point in the graph.
3. The current distribution on a selected wire can be plotted in amplitude, phase, real and imaginary parts vs. position in a 2D representation. The currents flowing on a structure can also be plotted as a color map on the wires.
4. Radiation and scattering fields are obtained, such as power density, directivity and gain patterns, total electric field, linearly and circularly polarized components, and Radar Cross Section (RCS). The surface-wave field can be obtained as a function of distance in the case of a real ground with finite conductivity.
5. The near-field components can be calculated in Cartesian, cylindrical and spherical coordinates. The field intensities can be plotted in 2D and 3D graphical representations and visualized as color maps in the proximity of a structure.
6. A 2D representation of radiated fields is available in Cartesian and polar coordinates.
7. 3D radiation patterns can be viewed with arbitrary viewing angles, zoom functions and colored mesh and surface, including a color bar-scale. 3D patterns can be plotted with specially designed lighting and illumination for an enhanced visualization of the simulation results.
8. Far-field patterns can be resolved into theta (vertical) and phi (horizontal) linearly polarized components, or right and left circularly polarized components.

9. The frequency spectrum of near- and far-fields can be seen in a 2D representation for all of the field components versus frequency.
10. An average radiated power test is performed for checking the accuracy of the simulation.
11. The computed data can be exported to .dat or .txt files to load the results in another software.
12. Suitable unit systems can be chosen for the plotted results (current scaling in KA, A, mA, uA; voltage scaling in KV, V, mV, uV; electric field scaling in KV/m, V/m, mV/m, uV/m; magnetic field scaling in KA/m, A/m, mA/m, uA/m; decibel scales, etc.).

1.2 Integrated Graphical Tools

AN-SOF® has a suite of integrated graphical tools for the convenient visualization of the simulation results. The following softwares are installed automatically and used by the main program:



AN-XY Chart®
This is a chart for plotting two related quantities, that is Y versus X. This tool permits plotting frequency-dependent quantities, such as current, voltage, impedance, reflection coefficient, VSWR, radiated power, consumed power, directivity, gain, radiation efficiency, radar cross section, etc. The current distribution on metallic structures can also be plotted as a function of position with this program. Besides, 2D radiation patterns can be represented for a near- or far-field as a function of a chosen angle or distance. Zoom with mouse support and several unit systems for the plotted results are available.

AN-Smith®
The famous Smith chart for the representation of impedances and admittances is implemented in this tool. An impedance/admittance curve in the Smith chart is obtained when frequency is varied. The frequency corresponding to each data point in the chart can easily be obtained by clicking with the mouse on the screen. Reflection coefficients and VSWR (Voltage Standing Wave Ratio) are also showed. Plots can be stored in independent files and opened later for a graphical analysis with AN-Smith®.

AN-Polar®
Radiation and scattering patterns versus azimuth or zenith angles can be represented in this polar diagram, from which the width of radiation lobes and front-to-back ratios can be obtained. The represented field quantities include power density, directivity, gain, normalized radiation pattern, total electric field, field polarized components, and radar cross section (RCS).

AN-3D Pattern®
A complete view of the radiation and scattering properties of a structure can only be achieved with a full-angle resolved pattern. This task can be accomplished with AN-3D Pattern®, which implements colored mesh and surface for the clear visualization of radiation lobes, including a color bar-scale indicating the field intensities over the lobes. Zoom, translation and rotation of the 3D pattern can be performed. The represented quantities include the power density, normalized radiation pattern, directivity, gain, total field, linearly polarized field components, circularly polarized field components, and RCS. Linear and decibels scales are available.

1.3 Intended Users

The main purpose of AN-SOF® is making simulation easy and accessible to a wide audience, so the software is designed for everyone interested in Electromagnetics and Electronics. No previous expertise in electromagnetic simulation is required to begin using this tool. AN-SOF® is frequently used by students, teachers, technicians, engineers, amateur radio hams, and everyone involved in the research and design of metallic antennas and passive circuits, from the very low frequency range to microwaves, as well as those dealing with radio engineering, microwaves, radar techniques, electromagnetic compatibility and communications.

The program can also be used for teaching purposes, primarily for research activities at the postgraduate level, but also for demonstration of antenna and scatterer phenomena.

1.4 Installing AN-SOF® and MCR®

AN-SOF® can be installed on a PC running Microsoft Windows® 98/XP/Vista/7 (32 and 64 bits). The minimum recommended hardware requirements are the following:

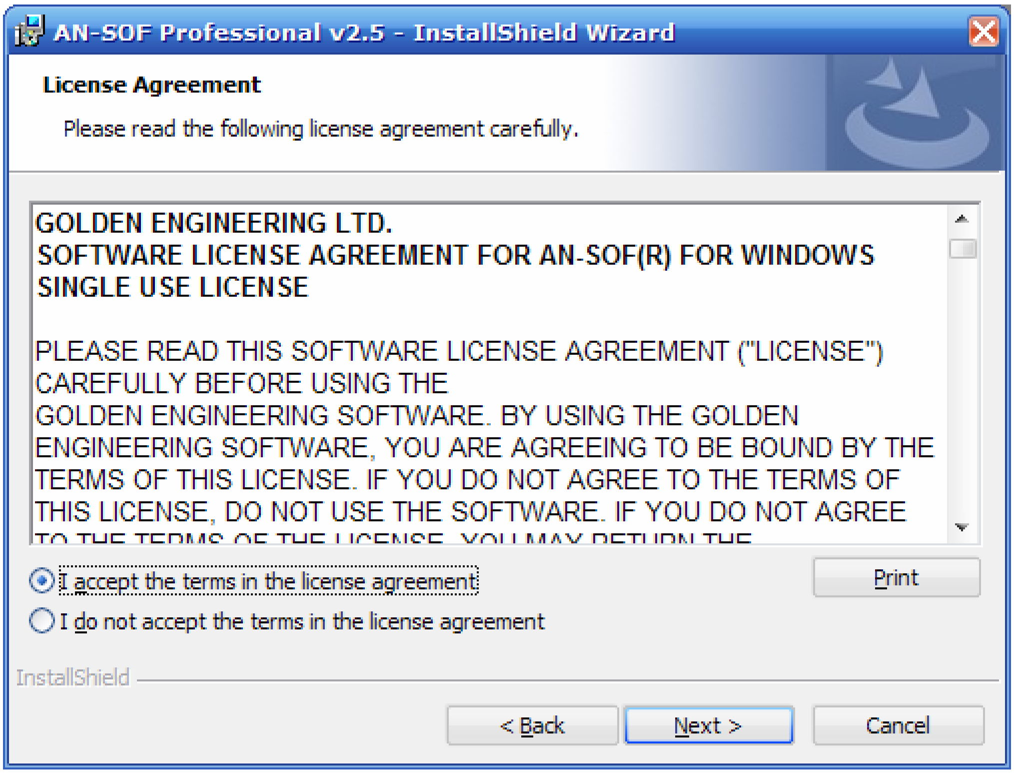
- 2 GHz processor.
- 500 MB free disk space.
- 512 MB RAM.
- VGA or super VGA graphics card (or equivalent).

The procedure for installing AN-SOF® is straightforward. Execute the SETUP.EXE program to install the software and follow the instructions on the screen. The **MATLAB® Component Runtime (MCR®)** must also be installed. This option will be shown once the AN-SOF® installation has finished, but can also be run from the Windows® Start Menu.

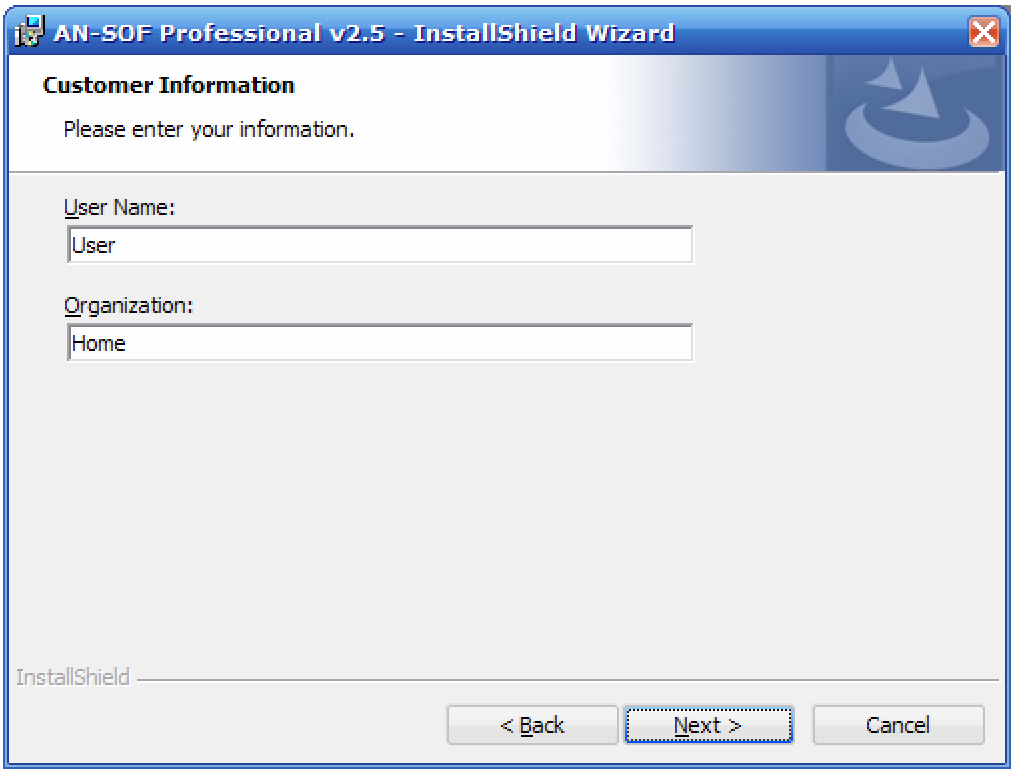
Please, follow these steps to install AN-SOF® and MCR®:

1. When the **AN-SOF®** installer startup screen appears, click **Next** to begin the installation.

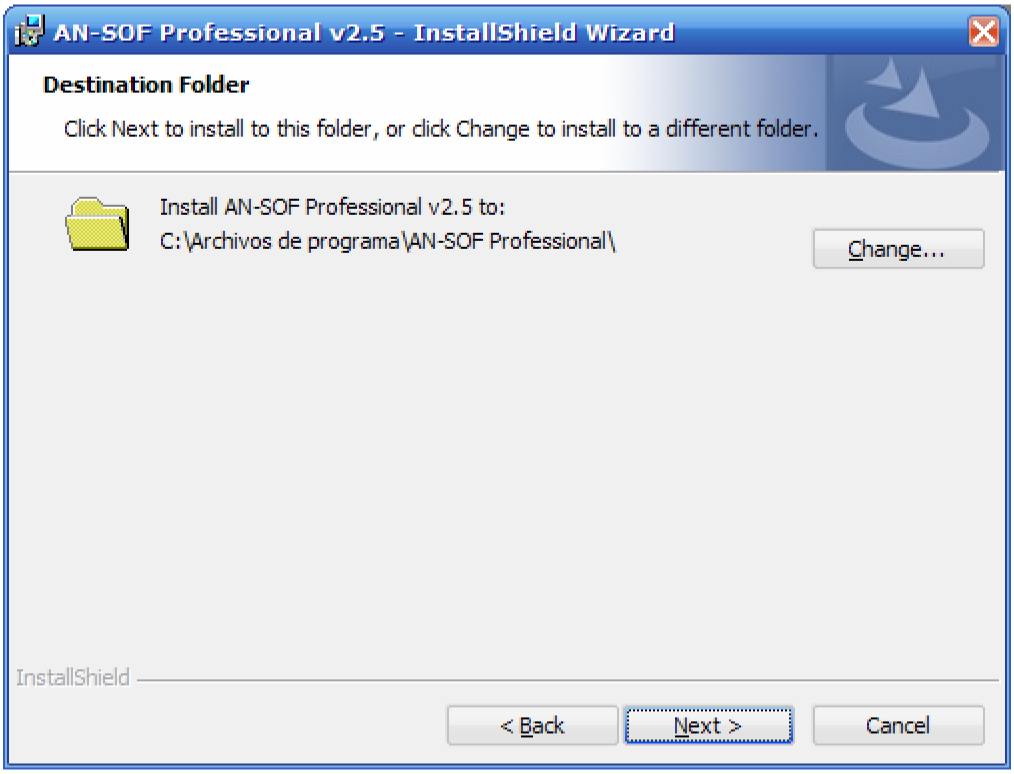
2. The setup wizard starts and the **license agreement** is shown. If you accept the terms in the license agreement, please click on this option and then press **Next**.



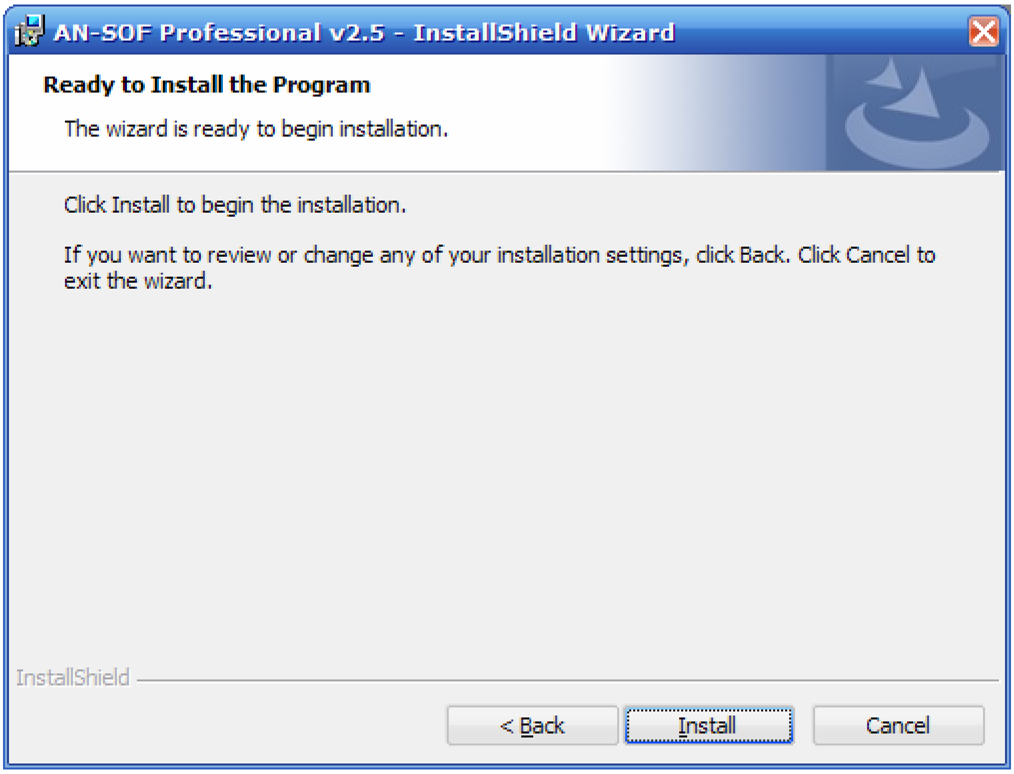
3. In the information screen a **user name** and **organization** can be entered.



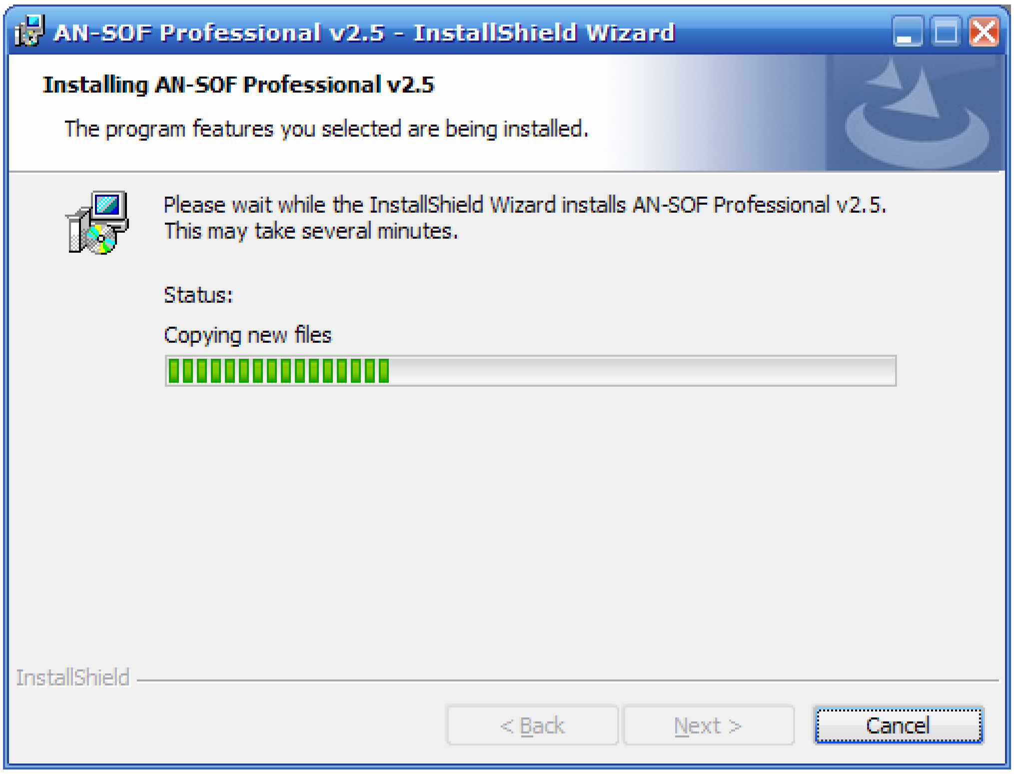
4. Choose the **destination folder** where AN-SOF® will be installed and click **Next** to continue.



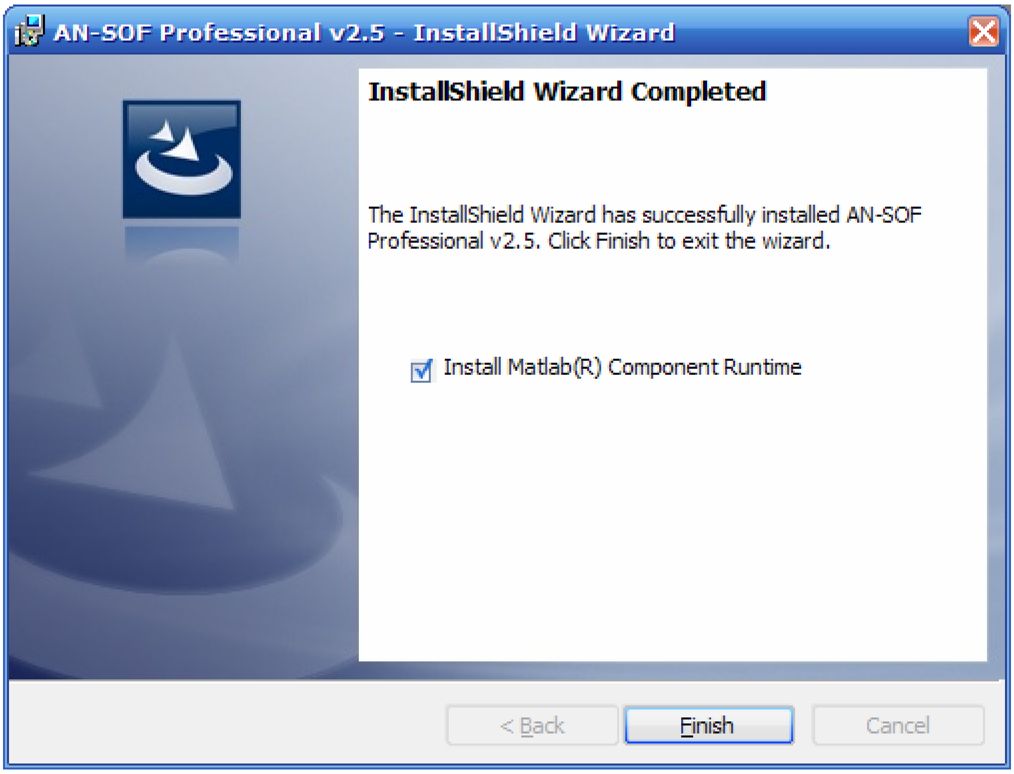
5. The wizard is ready to install AN-SOF®. Click **Install** to begin the process.



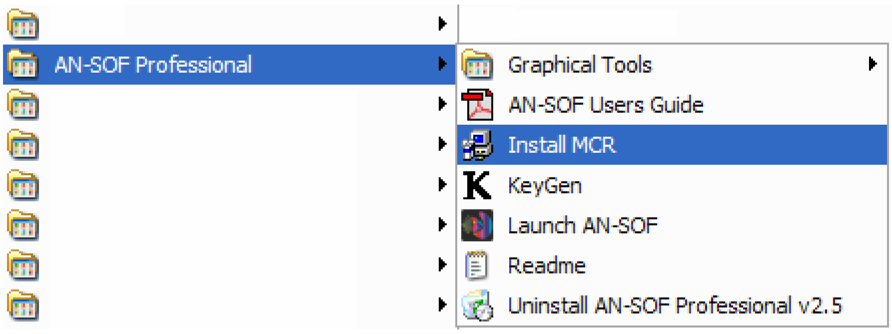
6. The installation begins.



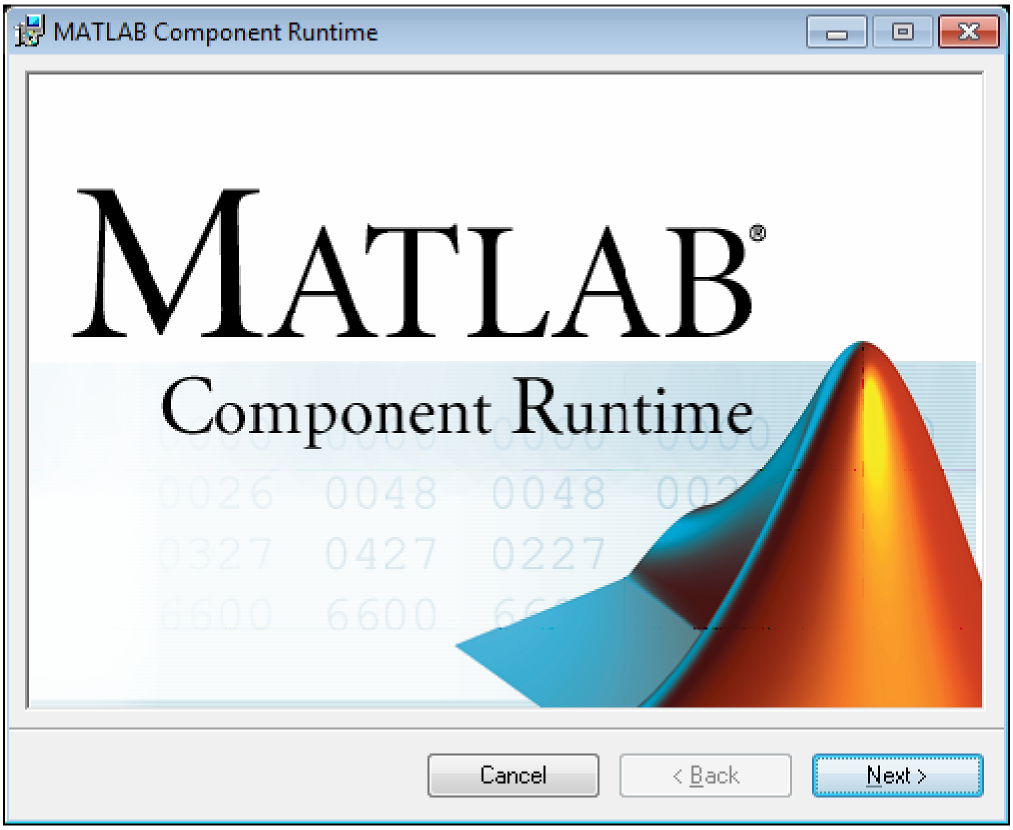
7. If AN-SOF® has successfully been installed, the **Install MATLAB(R) Component Runtime** option will be shown. Select this option and click **Finish** if you want to install the MCR now.



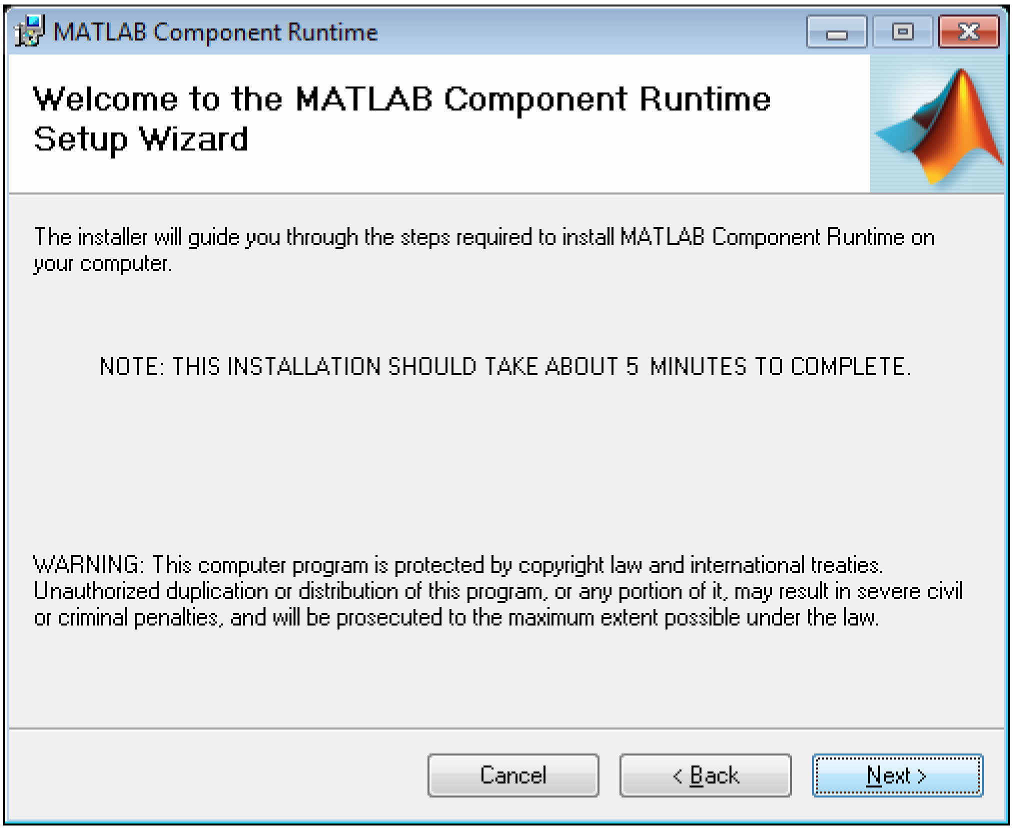
8. Once the AN-SOF® installation has finished, the MCR can also be installed by executing the **Install MCR** program located in the AN-SOF® folder within the Windows® Start Menu.



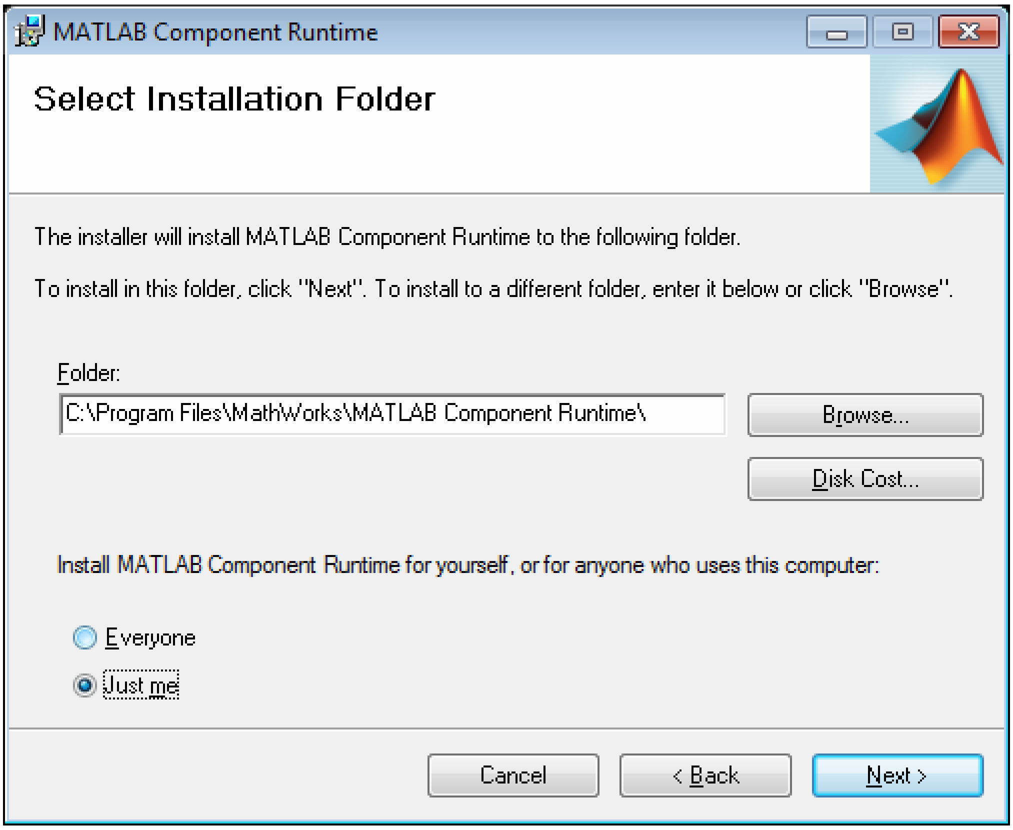
9. When the **MATLAB(R) Component Runtime** startup screen appears, click **Next** to begin the installation.



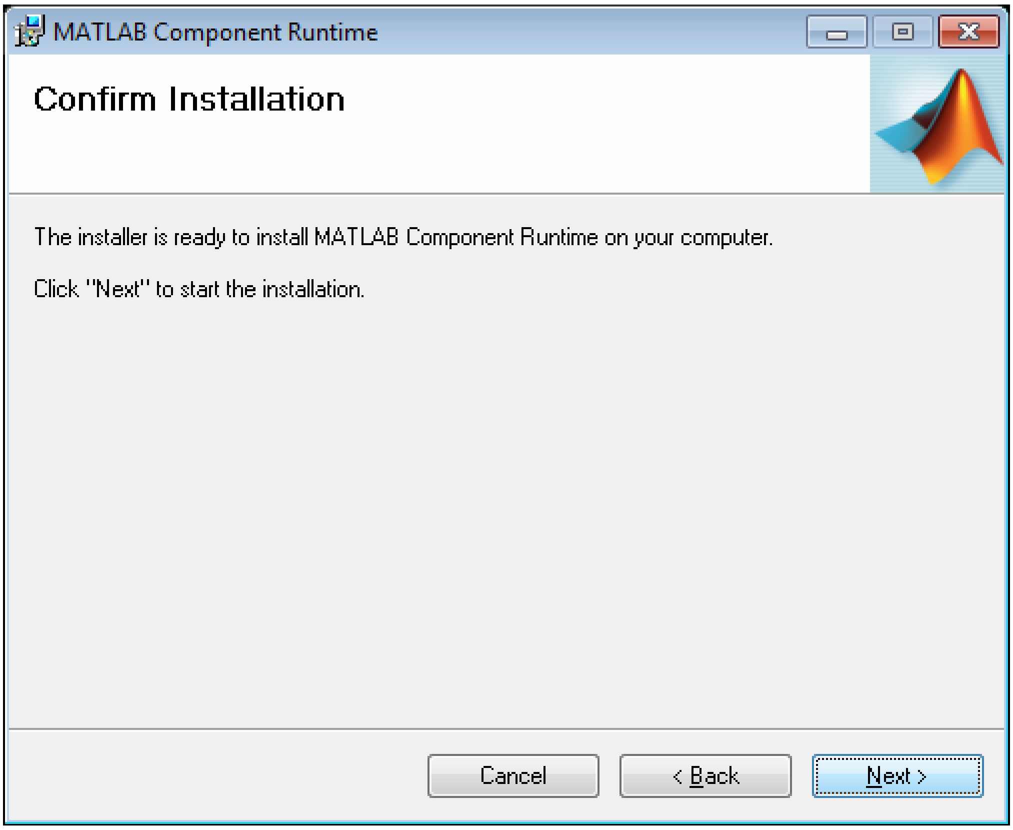
10. A welcome screen will be shown informing about installation time. Click **Next** to continue.



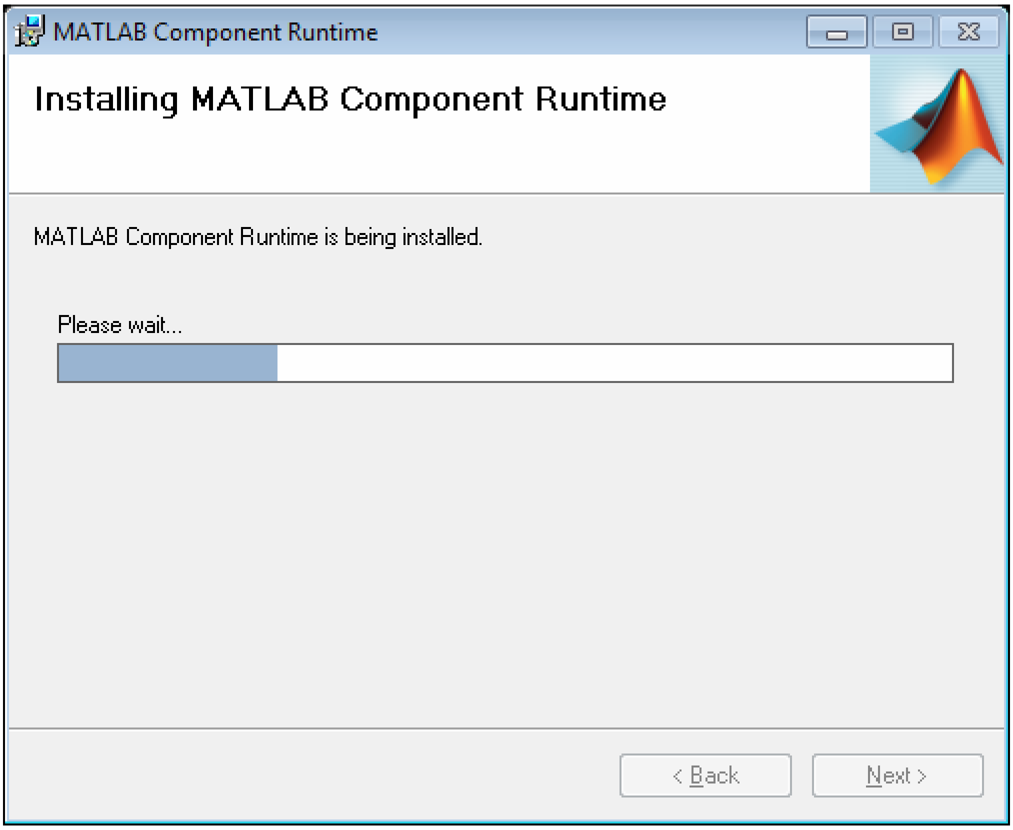
11. Choose the **installation folder** for the MCR and Click **Next**.



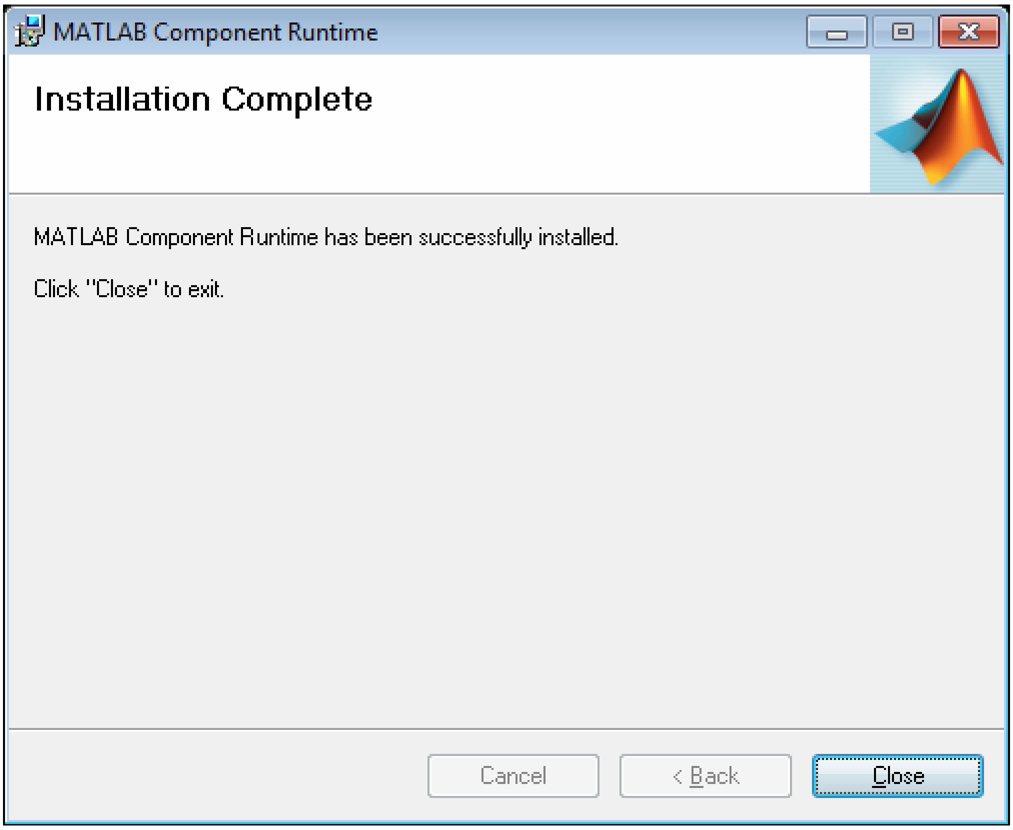
12. Confirm the installation by clicking **Next** to start the process.



13. The MCR installation begins.



14. Once the installation has finished, click **Close**.

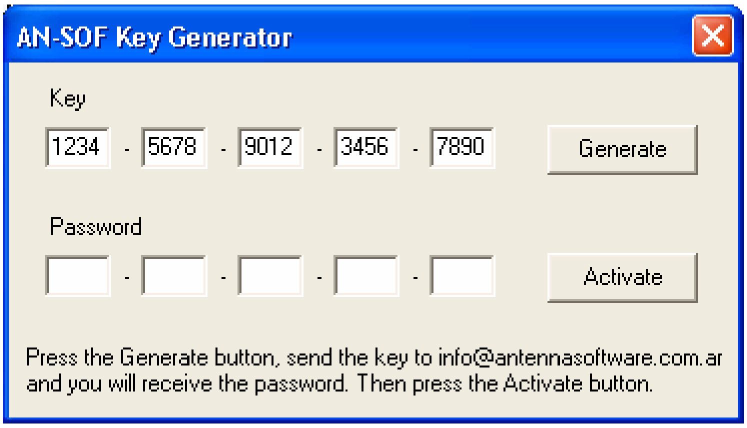


A folder with sample problem files, called EXAMPLES, will also be installed on the AN-SOF® installation directory.

1.5 Software Activation

A license key is provided per machine. Once AN-SOF® has been installed, it will be locked. In order to unlock the software, please perform the following steps:

1. Execute the Keygen® program. It is accessible from the AN-SOF® program folder on the Windows® Start menu.
2. In the Keygen® window, press the Generate button for generating a key number. This is a unique number per machine.



3. Please, send your key number to info@antennasoftware.com.ar and you will receive an activation password.
4. Copy the received password in the fields indicated in the Keygen® window and press the Activate button.
5. After this simple operation, you can begin to enjoy AN-SOF®.

1.6 AN-SOF® Versions

AN-SOF® v2.7 is provided in four versions:

AN-SOF100®
Full features, up to 100 wire segments for small sized structures.
:: Special Edition for evaluating the software capabilities ::

AN-SOF Basic®
Full features, up to 1000 wire segments for small and medium sized structures.
:: Special Edition for students and teachers ::

AN-SOF Classic®
Full features, up to 3000 wire segments for medium sized and relatively large structures.
:: Special Edition for getting started with electromagnetic simulation ::

AN-SOF Professional®
Full features, unlimited number of wire segments for large and very large structures.
:: Special Edition for electromagnetic professionals ::

Note: If you have installed AN-SOF100®, the software is free and it is ready to be used. If you are upgrading to a new version, please uninstall the previous versions of AN-SOF® and MCR® before installing the new one. The AN-SOF100® version must also be uninstalled before installing a Basic, Classic or Professional version.

1.7 New Features in AN-SOF® v2.7

Innovative new features have been included in this version. Among them, we can highlight the following:

Enhanced 3D visualization of fields and currents.

The 3D radiation lobes of far-field patterns can now be superimposed to the structure geometry to gain more insight into the directional properties of antennas and scatterers. Near-fields can also be shown as color maps in the proximity of antennas in three different representations: Cartesian, cylindrical and spherical. Besides, the currents flowing on a structure can be visualized directly on the wires as a colored intensity map.

Improved polar charts with beamwidth calculation.

The standard polar chart for vertical and horizontal slices of the 3D radiation pattern has been improved showing the maximum, -3dB and minimum radiation levels within the chart. The directions corresponding to these particular intensities as well as the beamwidth and front-to-back ratio are also calculated. Field values can be shown in the chart by mouse clicking on the polar lobes.

Fast and easy importation of wires from NEC-based files.

Besides the graphical input interface included in AN-SOF® for creating wires, text files containing geometrical data can be imported into the program. Three different file formats for importing wires are supported, including the still-in-use NEC (Numerical Electromagnetics Code) cards. With this feature, old antenna projects can be leveraged and updated.

2. Getting Started with AN-SOF®

2.1 Antenna Modeling Software

An antenna model is a representation of a real world antenna in a computer program. This kind of model should not be confused with a scale model that sometimes is built in order to measure the radiation characteristics of an identical antenna with a larger physical size. Due to the complexity of the math involved in a model, computer softwares are often programmed to predict and analize antenna performance.

An antenna modeling software can be used

- to learn more about antennas
- to get insight into the behavior of a particular antenna
- to design better antennas
- to predict antenna performance
- to tune for performance
- to try several possibilities before building the real model

Computer simulation in industry is used to overcome challenges and drive innovation in the product creation and development processes. A computer model has the advantage that it can be modified, redesigned, broken, destroyed and built again many times without wasting materials. Therefore, a considerable reduction in the cost of building successive physical models can be obtained during the design process with the help of a simulation software.

AN-SOF® is an antenna simulation software that allows us

- to describe the geometry of the antenna
- to choose construction materials
- to describe the environment and ground conditions
- to describe the antenna height above ground
- to analize the radiation pattern and front-to-back ratio
- to plot directivity and gain
- to analize impedance and SWR (Standing Wave Ratio)
- to predict bandwidth

and to get several more parameters and plots.

In order to plot the results from a simulation a suite of special programs, called AN-XY Chart®, AN-3D Pattern®, AN-Polar® and AN-Smith® have been included as integrated graphical tools. These tools can also be executed independently for a subsequent processing of graphics.

AN-SOF® is the easiest to use software for the simulation of antenna systems and, at the same time, it is the most accurate one. The key advantages can be summarized as follows:

- Fast and easy input and output graphical interfaces.
- Exact description of geometry details.
- Extended frequency range.
- MATLAB® Component Runtime for higher accuracy and speed.

2.2 Fundamentals of Simulation

AN-SOF® computes the electric currents flowing on metallic structures, including antennas in transmitting and receiving modes as well as scatterers. A scatterer is any object that can reflect and/or diffract radiofrequency waves. For example, the scattering of waves could be analized on the surface of an aircraft to investigate the best placement of an antenna, on a parabolic reflector to analize gain as a function of the reflector shape, on the chassis of a car to predict interference effects, etc.

One of the most validated methods for antenna simulation is the so-called *Method of Moments (MoM)*. An improved and advanced form of this method has been implemented in AN-SOF® to overcome various well-known difficulties of the traditional MoM.

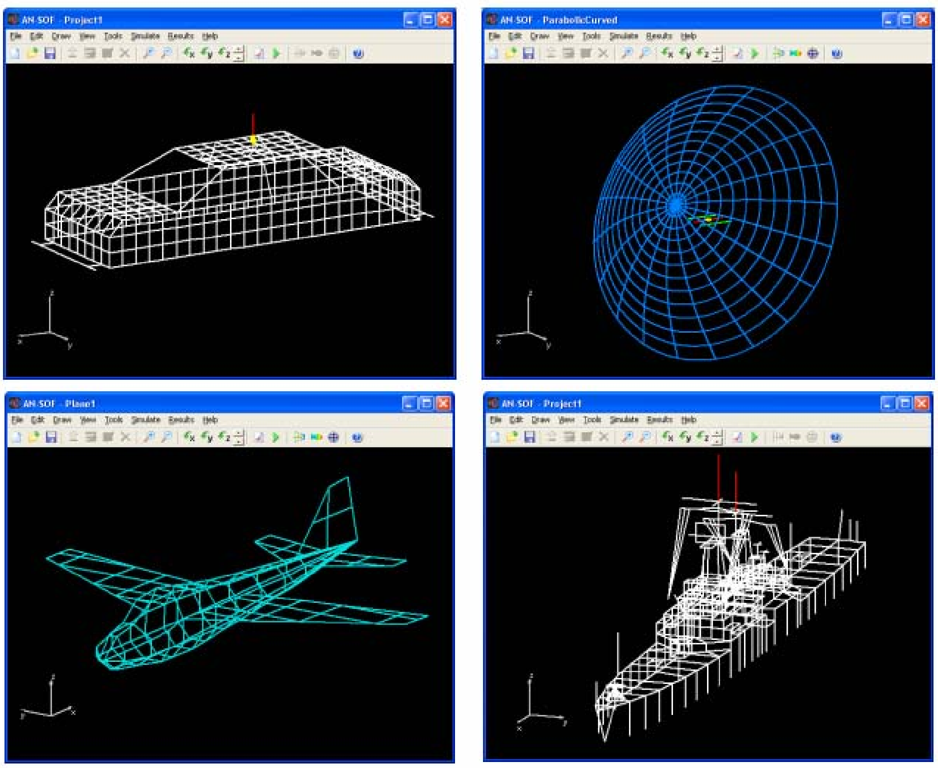


Fig. 2.1: Computer models of a car, a parabolic reflector, a plane and a ship using wire grids.

According to the MoM, any metallic structure can be modeled using conductive *wires*, as Fig. 2.1 shows. These wires must be divided into small pieces called *segments*. A wire segment has the shape of a cylindrical tube whose length should be short compared to the wavelength (λ) in order to get accurate results, Fig. 2.2. However, this is not a matter to worry about in a first simulation since automatic segmentation of wires can be defined in AN-SOF®. Electric currents can be forced to flow on the structure by placing a voltage generator at some position that works at a given frequency. Current generators can also be used as the excitation, as well as a plane wave impinging on the structure that comes from a far or distant source.

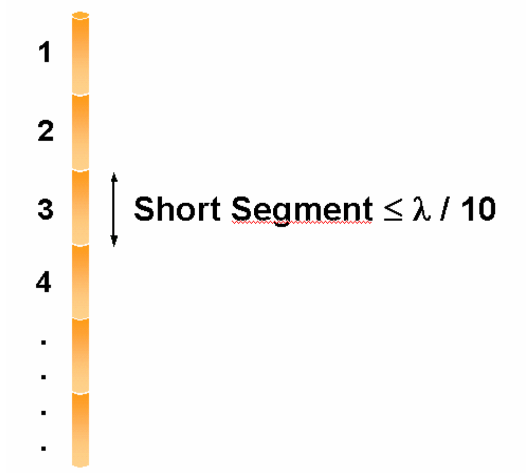


Fig. 2.2: A straight wire divided into short segments.

Once the structure geometry, materials and sources have been defined, the calculation can be run to obtain the currents flowing on the wire segments. In general, the electric currents will have varying intensities along and across the structure, so they are collectively referred to as a *current distribution*. Figure 2.3 shows an example of the current distribution on a log-periodic antenna.

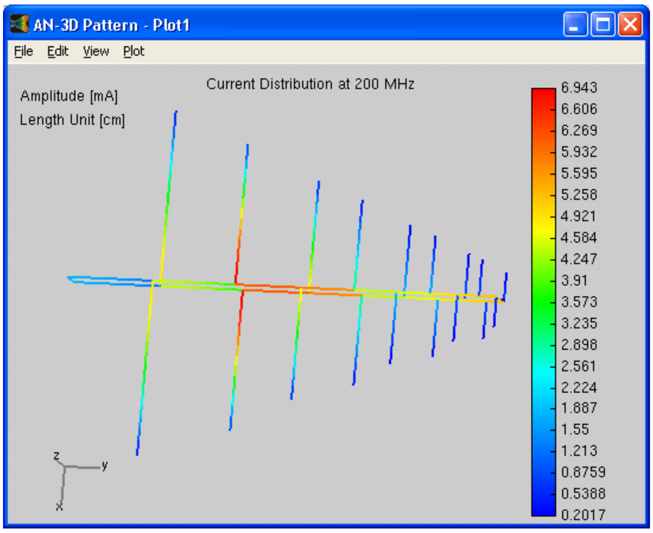


Fig. 2.3: Current distribution on a log-periodic antenna. The color map on the structure indicates the amplitudes of the electric currents.

The electromagnetic field radiated by the current distribution can be calculated in a second step of the simulation process. However, the current distribution itself gives a lot of information about the bevavior of the structure, specially if a frequency sweep has been performed. In the case of antennas, the feed point impedance can be obtained as a function of frequency to analize the bandwidth. The VSWR (Voltage Standing Wave Ratio) can be plotted in a Smith chart for a better interpretation of the results, Fig. 2.4.

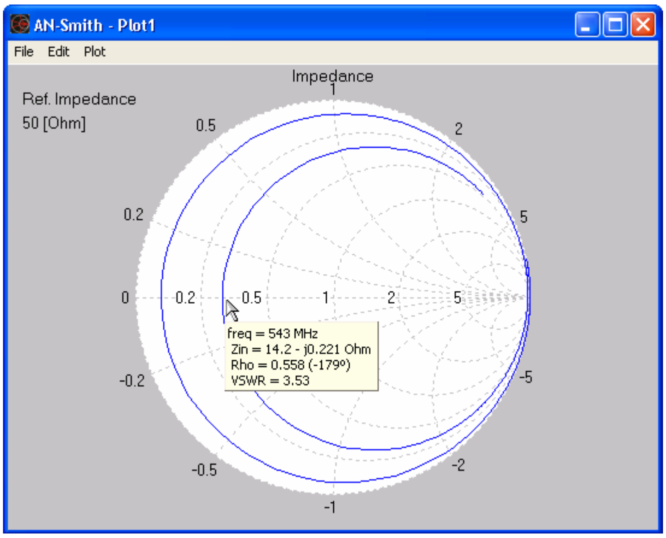


Fig. 2.4: Impedance plotted as a function of frequency in a Smith Chart, where the VSWR can be obtained by clicking on the curve.

The electric and magnetic fields can be obtained in the proximity of the structure, in the so-called *near-field zone*, and plotted as a color map whose intensities sometimes resemble the temperature maps in weather forecasts, Fig. 2.5.

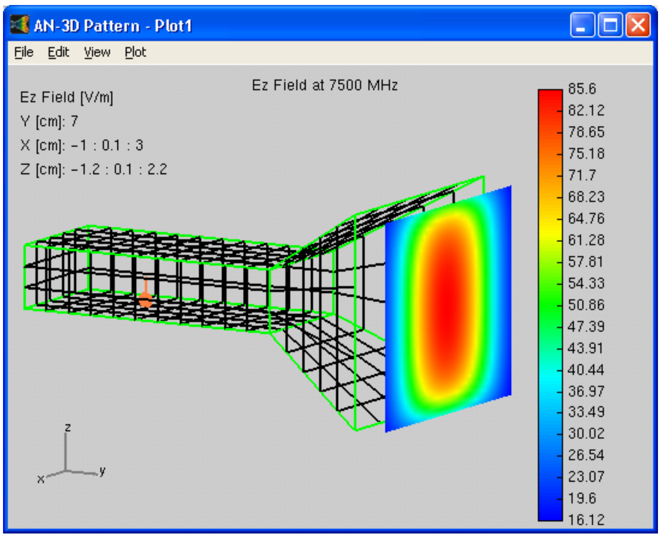


Fig. 2.5: Near electric field in the proximity of a Horn antenna.

Far away from the structure, at a distance of several wavelengths, the magnetic field becomes proportional to the electric field, so only the electric field intensities are often used to analize the results. This is the so-called *far-field zone*, where the radiated field is usually plotted as a function of direction in a polar diagram, Fig. 2.6. A more complete representation is obtained plotting a 3D pattern, where radiation lobes can be superimposed to the structure geometry for a better visualization of its directional properties, Fig. 2.7.

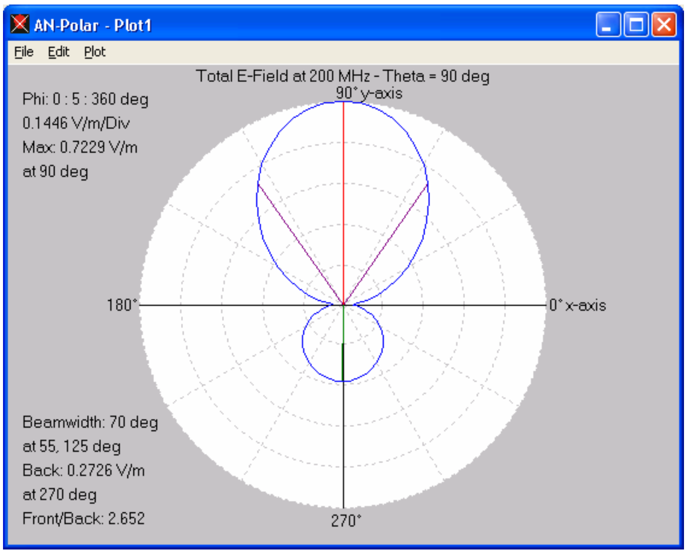


Fig. 2.6: Far-field pattern represented in a polar diagram. Beamwidth and front-to-back ratio are shown.

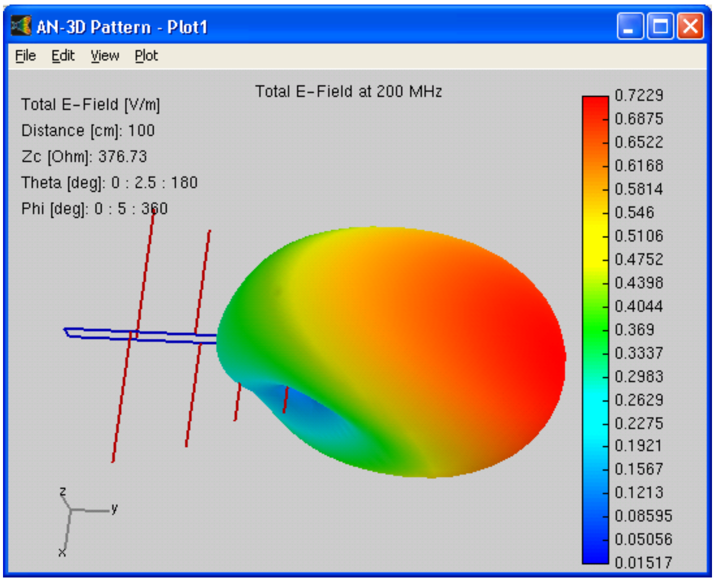


Fig. 2.7: Far-field pattern represented in a 3D plot and superimposed to the antenna geometry.

In summary, simulating a wire structure is a three-step procedure:

1. **Defining the frequency, geometry, materials and sources.**
2. **Running the calculation.**
3. **Visualizing the computed results.**

A convenient unit system for the frequencies and lengths can be chosen at the beginning of the simulation and can then be changed at any time. For example, the wire lengths are often measured either in meters (m) or feet (ft) at frequencies below 100 MHz, while either millimeters (mm) or inches (in) are preferred at higher frequencies. The wire geometry can easily be drawn on the screen using dialog boxes for the input data. The structure is placed in 3D space where a set of 3D-tools have been implemented for a better user experience.

2.3 The Conformal Method of Moments

In the traditional Method of Moments (MoM) the structures to be modeled are divided into straight wire segments. Straight segments fit well the shape of linear antennas like dipoles and arrays constructed using dipoles. However, there are many antennas and structures that have curved shapes. In these cases, a curved wire is approximated using a string of straight-line segments, Fig. 2.8(a). Sharp junctions between adjacent wires introduce a modeling error at the very beginning of the simulation that can never be fixed. Poor results for curved antennas like loops, helices and spirals are often obtained when the linear approximation is applied, especially large errors in the feed point impedances.

Another problem in the traditional MoM arise for wires bent at right angles and for angles less than 30 degrees between adjacent wires, Fig. 2.8(b). In these cases, a lot of segments are needed near the wire corners to get reliable results.

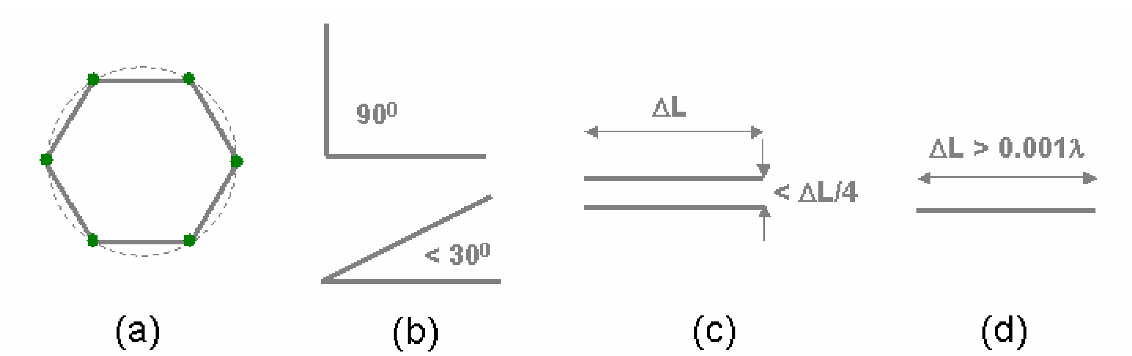


Fig. 2.8: Limitations of the traditional Method of Moments.

A third problem that should be pointed out is about the distance between parallel wires. Segments cannot be very close to each other since misleading results are obtained when the separation between them is less than a quarter of a segment length, Fig. 2.8(c).

The segment length itself has a limitation, it must be greater than 0.001 of a wavelength, and consequently the traditional MoM cannot be applied at very low frequencies, Fig. 2.8(d). For example, consider an electric circuit around 1 meter in size operating at 60 Hz. The wavelength (in free space) can be calculated as (300/60)x1,000,000 = 5,000,000 meters. Thus, the size of the circuit measured in wavelengths is 1/5,000,000 = 0.0000002, so segments shorter than 0.0000002 of a wavelength are needed to model the circuit. This segment length is at least 5,000 times shorter than the minimum segment length supported by the MoM. Therefore, an electric circuit at low frequencies cannot be modeled using the traditional implementation of the MoM for wire antennas.

The limitations of the traditional MoM have been removed in its improved version: the *Conformal Method of Moments (CMoM)*. In the CMoM, *conformal segments* are used that exactly follow the contour of the structure, so an exact description of geometry details is achieved, Fig2.9. A conformal segment is a curved cylindrical tube that fit correctly the shape of curved wires. The limitations regarding bent wires and small separations between wires have been removed by means of improved numerical techniques for solving integrals and matrix equations via the MATLAB® Component Runtime (MCR). The MCR is a math library that has been integrated to AN-SOF® for performing high accuracy calculations.

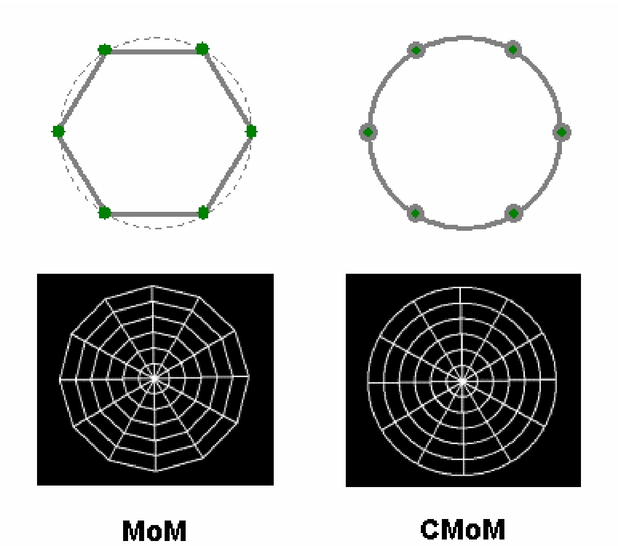


Fig. 2.9: A circular loop and a disc modeled using the traditional MoM and the Conformal MoM.

AN-SOF® is the only antenna modeling software that implements the CMoM. The advantages of the CMoM can be summarized as follows:

- Using curved segments the number of calculations is reduced and accuracy is greatly increased.

- Simulation time and computer memory space are reduced, allowing for the solution of bigger problems.

- Advanced calculation techniques make possible simulations from extremely low frequencies (e.g. electric circuits at 50-60 Hz) to very high ones (microwave antennas above 1 GHz). This extended frequency range is only available in AN-SOF®.

2.4 Performing the First Simulation

Several example files are included in the AN-SOF® installation directory within a folder named EXAMPLES. Opening a file with extension .emm will show the wire structure on the screen. The calculation can be run by pressing the Run ALL button (F5) in the AN-SOF® toolbar. The main results can be plotted by pressing the Plot Current Distribution button (F8), the Far-Field 3D Plot button (F9) and the Far-Field Polar Plot button (Ctrl+F9).

As a first experience using AN-SOF®, a simulation of a standard half-wave dipole could be performed since this is one of the simplest antennas that can be modeled. A dipole is just a straight wire fed at its center. When the wire cross-section is circular, the dipole is called a *cylindrical antenna*. Since the material the wire is made of is usually a very good conductor, the wire can be considered to be a *perfect conductor*, that is, a material that has zero resistivity. Therefore, a cylindrical antenna with zero resistivity will be modeled in this example.

The first step is to set the operating frequency. Go to *Simulate* in the main menu and choose *Configure*. The *Configuration* dialog box will appear. In the *Frequency* tab there are three options. Select *Single* and then write the operating frequency for the antenna, Fig. 2.10. In this case, frequency is given in megahertz (MHz) and lengths are measured in meters (m). Go to section 3.3 (Setting up Preferences) to change the unit system for frequencies and lengths if desired. Please, note that for a frequency of 300 MHz, the wavelength equals 1 meter.

After defining the operating frequency, the antenna geometry can be drawn on the workspace. The workspace is the place on the screen where the wire structure is drawn and it represents the 3D space where the structure can be zoomed, rotated, and moved.

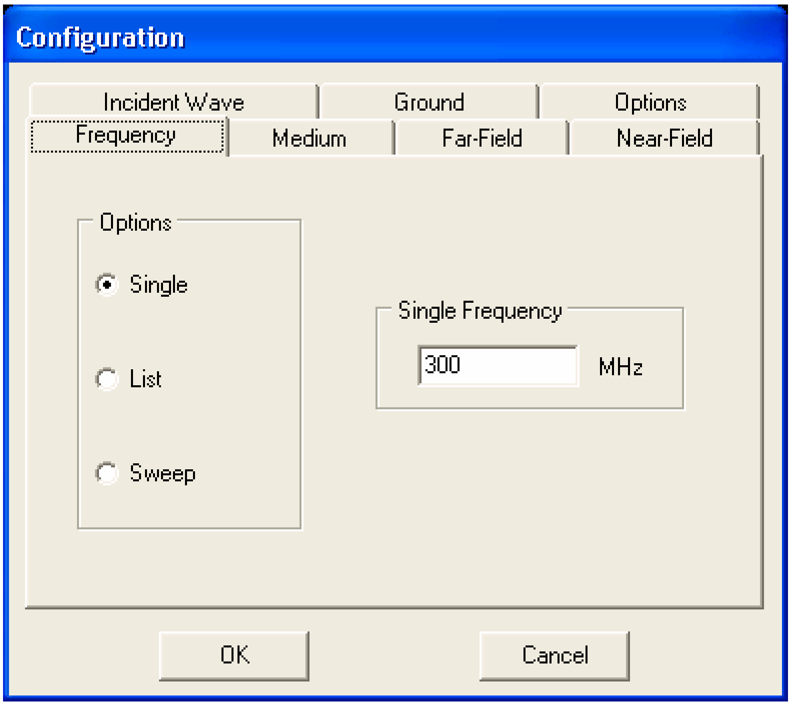


Fig. 2.10: Frequency tab in the Configuration dialog box where a frecuency of 300 MHz is defined.

A straight wire is called a *Line* in AN-SOF®. Go to *Draw/Wire/Lline* in the main menu. The *Draw* dialog box will be shown. In the *Line* tab, the coordinates of two distinct points can be defined. In this example, the line will be along the z-axis and will be 0.5 meters long, which corresponds to half a wavelength at 300 MHz. Figure 2.11 shows that the starting point of the line is chosen at (X1,Y1,Z1) = (0,0,-0.25) while the ending point is at (X2,Y2,Z2) = (0,0,0.25).

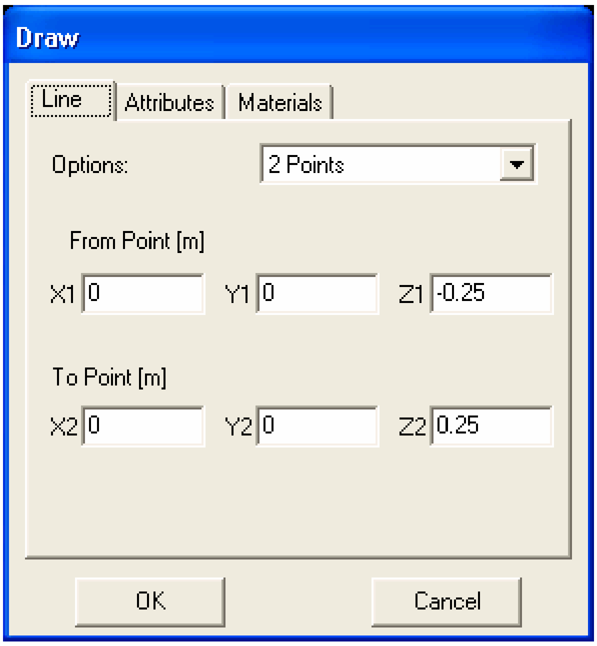


Fig. 2.11: Line tab in the Draw dialog box for defining a straight line.

Then, click on the *Attributes* tab, Fig. 2.12. The line must be divided into segments, which must be short compared to the wavelength. Basically, if the segment length is equal or less than a tenth of a wavelength, it is considered to be a short segment. AN-SOF® automatically suggests a minimum number of segments to achieve reliable results. To get more resolution, the number of segments can be increased. In this case, the line will be divided into 17 segments. The wire cross-section will be circular with 5 millimeters in radius.

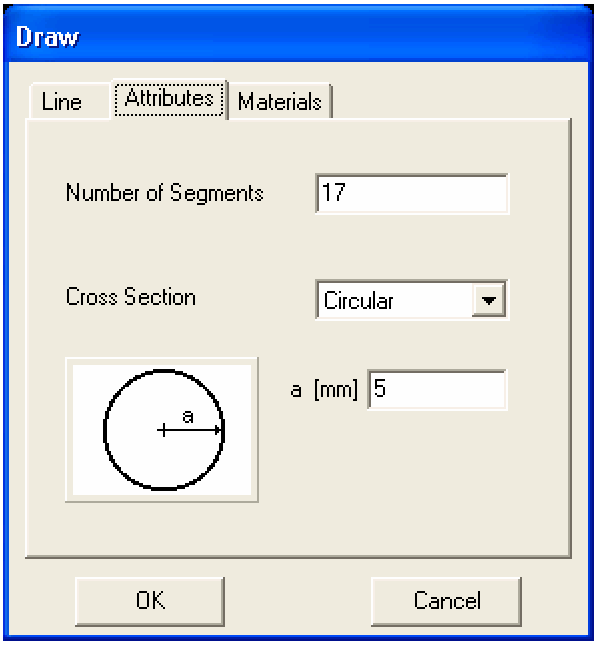


Fig. 2.12: Attributes tab in the Draw dialog box for setting the number of segments and wire radius.

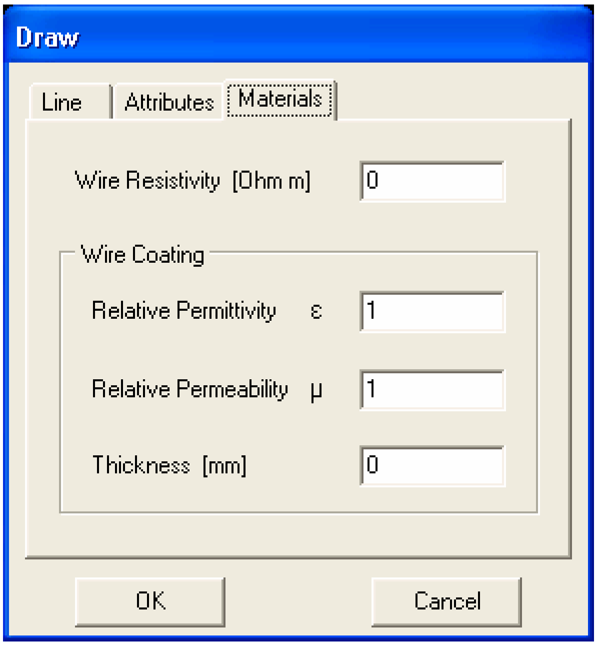


Fig. 2.13: Materials tab in the Draw dialog box for setting the wire resistivity.

In the *Materials* tab the wire resistivity will be set to zero, as it has been mentioned, Fig. 2.13.

The next step is to feed the dipole antenna. Click with the right mouse button on the wire. Choose the *Source-Load* command from the pop-up menu. The *Source-Load* toolbar will be displayed. Move the track-bar cursor to the center of the wire. Then, press the *Add Source* button. A voltage source 1 Volt in amplitude and zero phase is defined, Fig. 2.14.

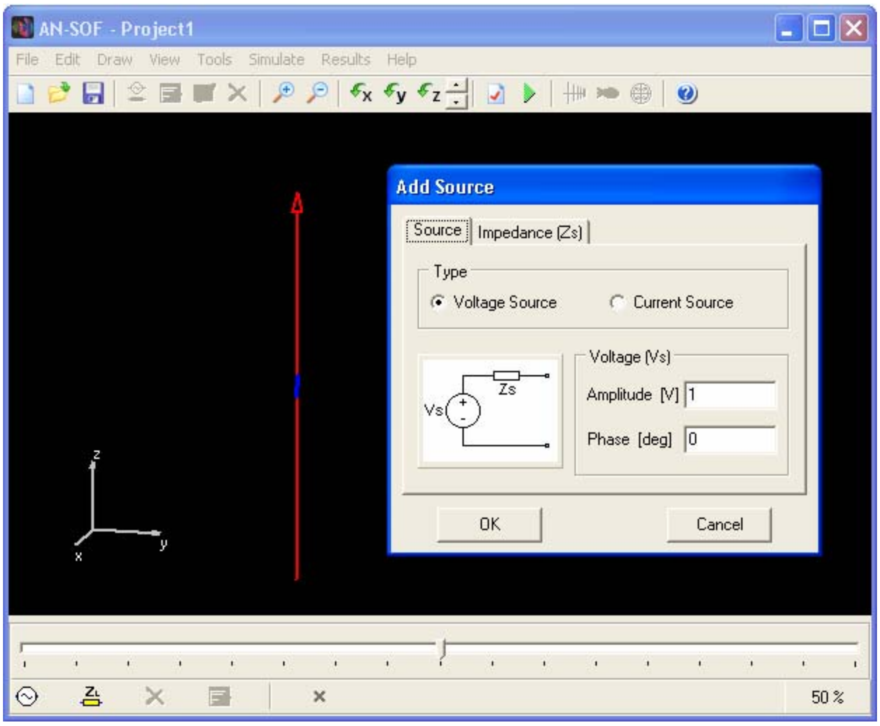


Fig. 2.14: Add Source dialog box shown after pressing the Add Source button in the Source-Load toolbar.

The simulation can be run by pressing *Simulate/Run Currents* in the main menu. Once the calculation has finished, press *Simulate/Run Far-Field*. In this way, the current distribution on the dipole antenna and the radiation field have been computed.

AN-SOF® has integrated graphical tools for the convenient visualization of the simulation results. Click on the wire with the right mouse button and select *Plot Currents* in the pop-up menu. A plot of the current distribution in amplitude along the dipole antenna will be shown, Fig. 2.15. Since a half-wave dipole has been defined, the resulting current distribution is a semi-cycle approaching a sine function.

Several parameters from the point of view of the voltage source connected to the antenna can be obtained. Click on the wire with the right mouse button and select *List Currents* in the pop-up menu. Move the track-bar cursor to the position of the voltage source and press the *Input List* button. The input impedance of the dipole antenna will be shown and many other parameters, Fig. 2.16.

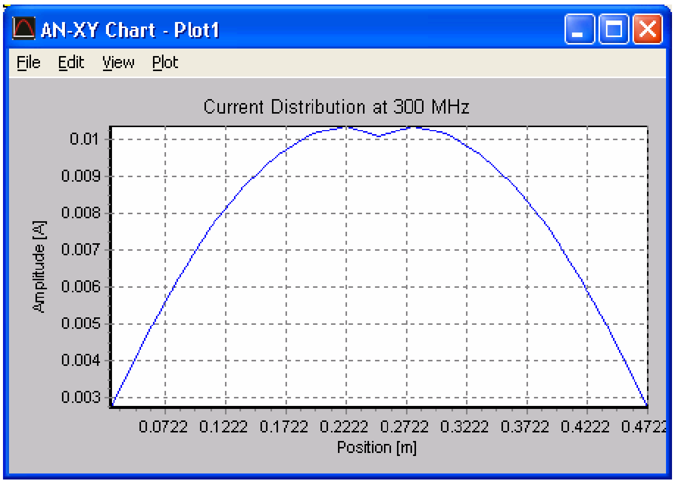


Fig. 2.15: Current distribution along a half-wave dipole.

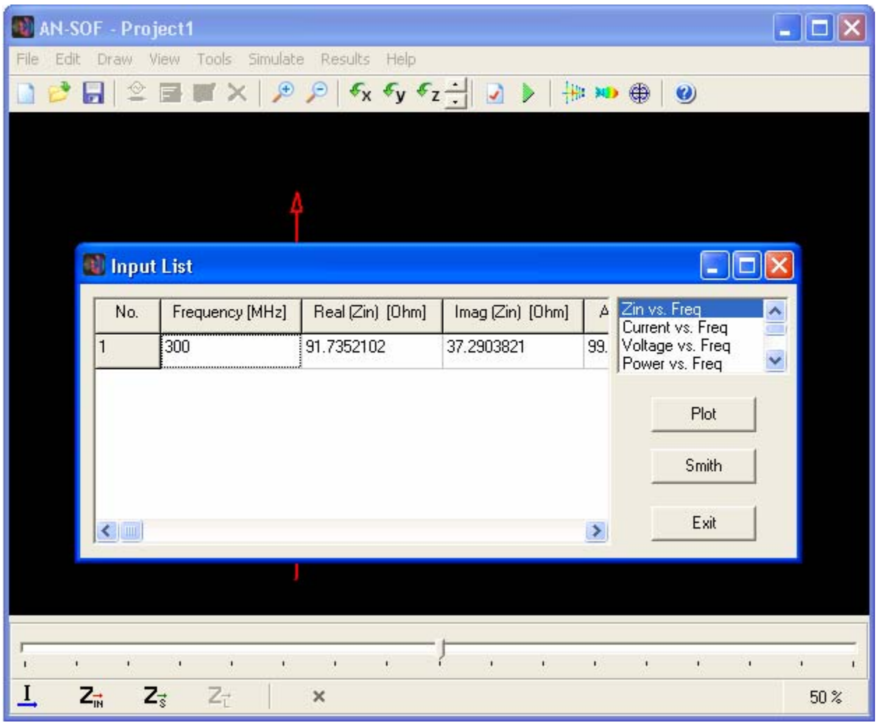


Fig. 2.16: Input List dialog box where the input impedance can be seen.

The radiation pattern can be represented in a 3D plot. Choose *Results/Plot Far-Field/3D Plot* in the main menu. The 3D power density pattern will be displayed. A color bar-scale indicates the field intensities over the radiation lobes. The directivity, gain and field patterns can also be plotted. It can be seen that the half-wave dipole is an omnidirectional antenna in the plane perpendicular to the dipole axis, Fig. 2.17.

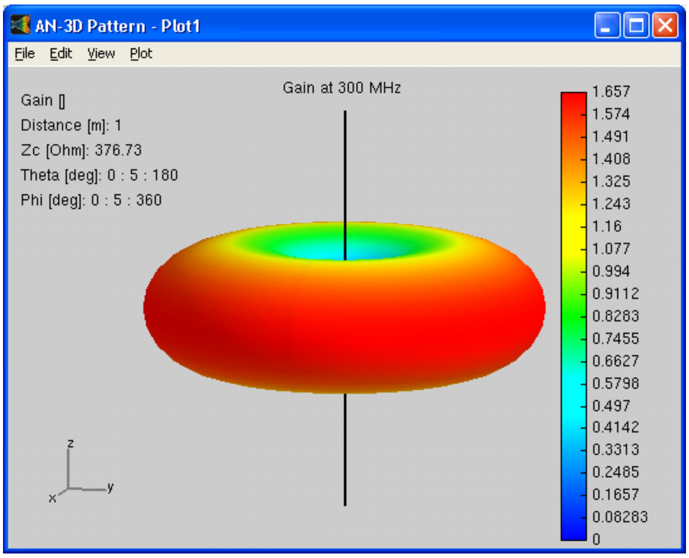


Fig. 2.17: Gain pattern of a half-wave dipole.

The following sections of this User's Guide describes AN-SOF® and its many functions in detail. The guide is organized according to the steps that should be followed when performing a standard simulation and explains all aspects of using AN-SOF® in detail. Technical assistance and support can be requested via e-mail to support@antennasoftware.com.ar.

3. The AN-SOF® Interface

When AN-SOF® is started, the initial screen contains the following components:

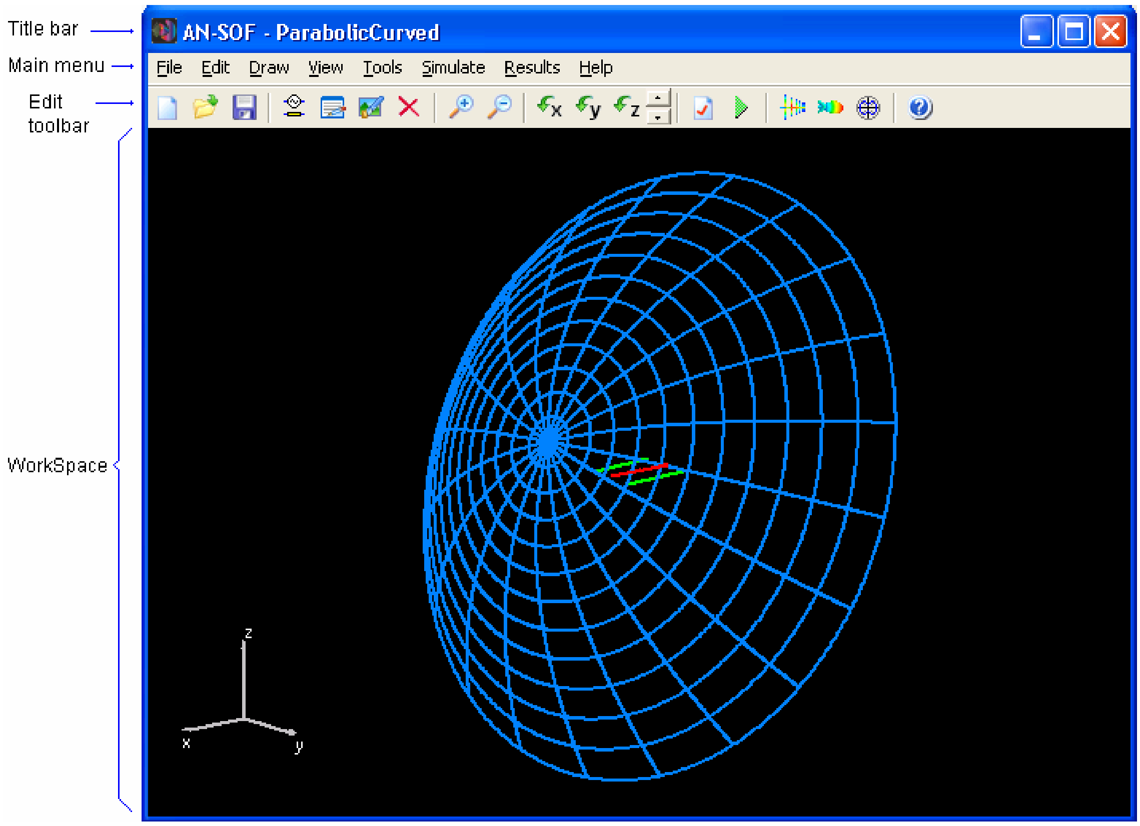


Fig. 3.1: The AN-SOF® interface.

- The **Title bar** contains the name of the currently active project (.EMM file).
- The **Main Menu** bar contains the File, Edit, Draw, View, Tools, Simulate, Results and Help menus.
- The **Edit toolbar** contains icons that represent commands.
- The **Workspace** is the place on the screen where the wire structure is drawn. It represents the 3D space where the structure can be zoomed, rotated and moved. The Workspace background can be black or white.

3.1 Main Menu

The Main Menu bar contains the following menus:

File Menu

Use the File menu to open, save, close, and print new or existing projects. This menu has the following commands:

New... (Ctrl+N)
Creates a new project.

Open... (Ctrl+O)
Displays the Open dialog box for loading an existing project (.EMM file).

Save (Ctrl+S)
Saves the currently active project using its current name.

Save As...
Saves the currently active project using a new name. Also saves a new project using a name specified by the user.

Import Wires
Displays the Import dialog box for importing a list of wires in NEC, MM and EZ formats.

Copy Workspace
Sends the project workspace to the clipboard.

Print...
Sends the project workspace to the printer.

Exit (Ctrl+Q)
Closes the open project and then exits AN-SOF®.

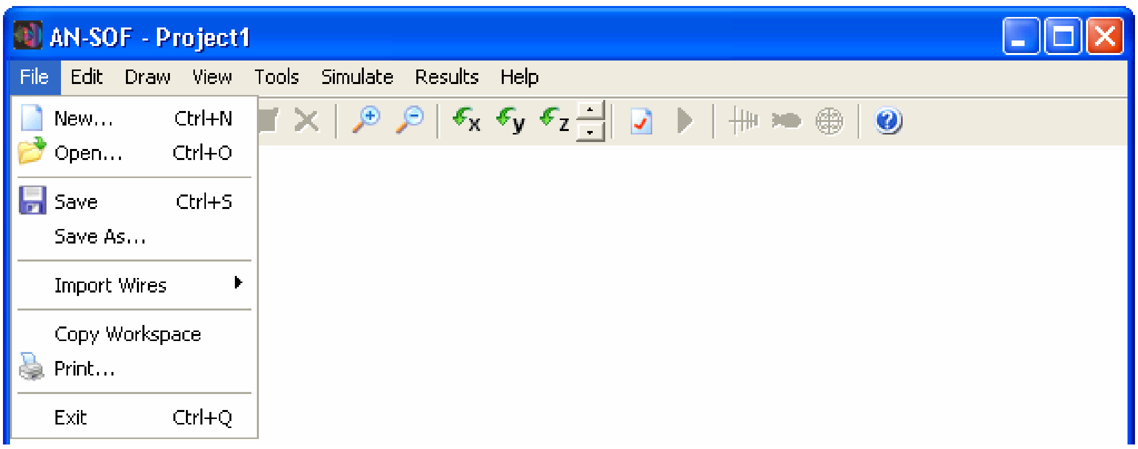


Fig. 3.2: File menu.

Edit Menu

Use the Edit menu commands to edit and handle wires and wire grids.
This menu has the following commands:

Undo (Ctrl+Z)
Returns the project to the status before a command was executed.

Source/Load (Ins)
Displays the Source/Load toolbar for exciting or loading the selected wire. This command is enabled when a wire is selected.

Modify (Ctrl+Ins)
Displays the Modify dialog box for modifying the selected wire.
This command is enabled when a wire is selected.

Wire Color
Displays a Windows® dialog box for changing the color of the selected wire structure. This command is enabled when a wire structure is selected.

Delete (Del)
Deletes the selected wire, wire grid or group of wires with all sources and loads placed on it. This command is enabled when a wire, wire grid or group of wires is selected.

Preferences
Displays a Windows® dialog box for setting up the preferred options for unit systems, workspace color and confirmation questions.

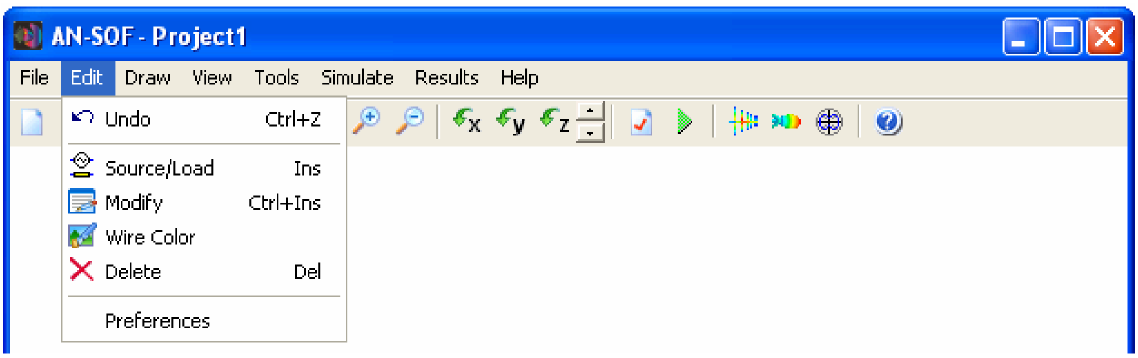


Fig. 3.3: Edit menu.

Draw Menu

Use the Draw menu commands to create and draw wires and wire grids.
This menu has the following commands:

Wire
Creates and draws a new wire on the project workspace. This option has a sub-menu with the following commands:

Line
Opens the Line dialog box for drawing a line or straight wire.

Arc
Opens the Arc dialog box for drawing an arc or arced wire.

Circle
Opens the Circle dialog box for drawing a circle or circular loop.

Helix
Opens the Helix dialog box for drawing a helix or helical wire.

Quadratic
Opens the Quadratic dialog box for drawing a quadratic wire.

Archimedean Spiral
Opens the Archimedean Spiral dialog box for drawing an Archimedean spiral.

Logarithmic Spiral
Opens the Logarithmic Spiral dialog box for drawing a logarithmic spiral.

Wire Grid
Creates and draws a new wire grid on the project workspace. This option has a sub-menu with the following commands:

Plate
Opens the Draw dialog box for drawing a plate or bilinear surface.

Disc
Opens the Draw dialog box for drawing a disc.

Flat Ring
Opens the Draw dialog box for drawing a flat ring or a disc with a hole at its center.

Cone
Opens the Draw dialog box for drawing a cone.

Truncated Cone
Opens the Draw dialog box for drawing a truncated cone.

Cylinder
Opens the Draw dialog box for drawing a cylinder.

Sphere
Opens the Draw dialog box for drawing a sphere.

Paraboloid
Opens the Draw dialog box for drawing a parabolic surface.

Tapered Wire
Creates and draws a new tapered wire on the project workspace. This option has a sub-menu with the same commands as the Wire option described above, but each wire can have a stepped radius along its length.

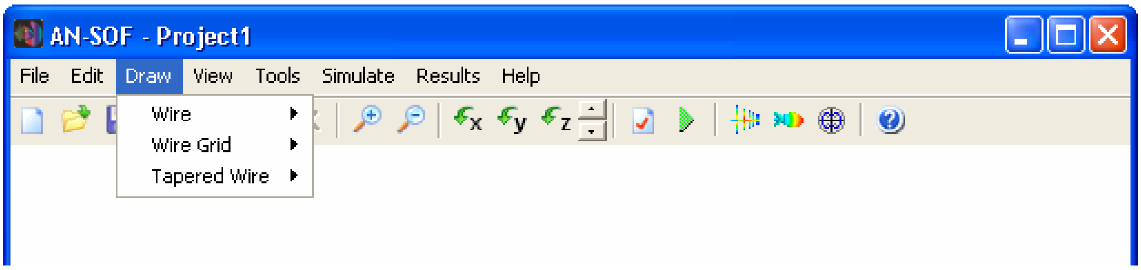


Fig. 3.4: Draw menu.

View Menu

Use the View menu commands to display or hide different elements of the AN-SOF® environment, zoom the wire structure and view additional information about the project and wires.
This menu has the following commands:

Wire Properties... (Ctrl+W)
Displays the Wire Properties dialog box for viewing information about the selected wire. This command is enabled when a wire is selected.

Project Details...
Displays the Project Details dialog box for viewing information about the currently active project.

Zoom In (+)
Increases the size of the wire structure on the workspace.

Zoom Out (-)
Decreases the size of the wire structure on the workspace.

Axes (Ctrl+A)
Displays the Axes dialog box for changing the appearance of the axes on the project workspace.

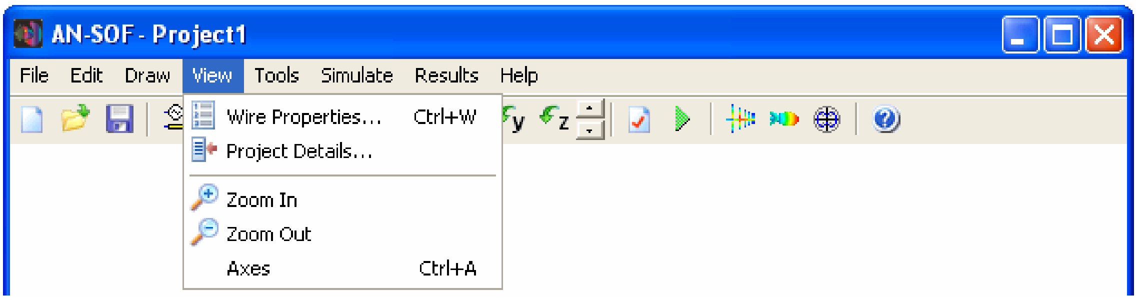


Fig. 3.5: View menu.

Tools Menu

Use the Tools menu commands to display 3D, polar, rectangular, and smith charts. This menu has the following commands:

3D Chart
Executes the AN-3D Pattern® program for opening 3D plot files (.P3D).

Polar Chart
Executes the AN-Polar® program for opening polar plot files (.PLR).

Rectangular Chart
Executes the AN-XY Chart® program for opening rectangular plot files (.PLT).

Smith Chart
Executes the AN-Smith® program for opening Smith plot files (.STH).

Calculator
Executes the Microsoft Windows® Calculator.

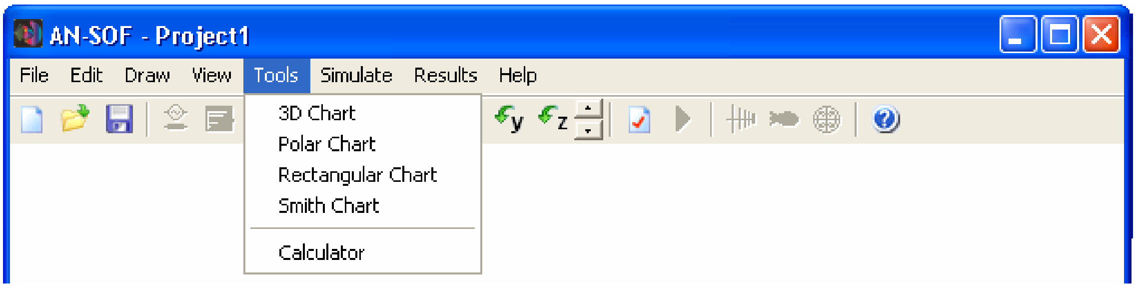


Fig. 3.6: Tools menu.

Simulate Menu

Use the Simulate menu commands to configure and run the simulation.
This menu has the following commands:

Configure... (Ctrl+C)
Displays the Configuration dialog box for defining the type of simulation, the operating frequencies of the system and other several options.

Run ALL (F5)
Runs the computation of the current distribution, far- and near-fields.

Run Currents and Far-Field (F6)
Runs the computation of the current distribution and far-fields.

Run Currents and Near-Field (F7)
Runs the computation of the current distribution and near electric and magnetic fields.

Run Currents
Runs the computation of the current distribution on the wire structure.
This command is disabled when the currents are computed.

Run Far-Field
Runs the computation of the far-field generated by the currents flowing on the wire structure.
This command is enabled when the currents are computed.

Run Near E-Field
Runs the computation of the near electric field generated by the currents flowing on the wire structure.
This command is enabled when the currents are computed.

Run Near H-Field
Runs the computation of the near magnetic field generated by the currents flowing on the wire structure.
This command is enabled when the currents are computed.

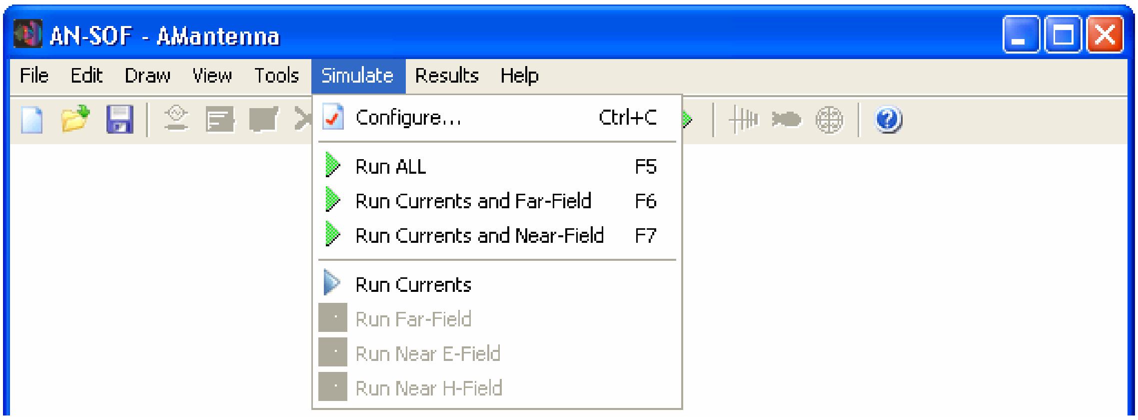


Fig. 3.7: Simulate menu. Run Currents command enabled.

Results Menu

Use the Results menu commands to visualize the results from a simulation. This menu has the following commands:

Plot Current Distribution (F8)
Executes the AN-3D Pattern® program for plotting the current distribution as a colored pattern over the wire structure.

Plot Currents (Ctrl+F8)
Executes the AN-XY Chart® program for plotting the currents vs. position along the selected wire. This command is enabled when a wire is selected.

List Currents...
Displays the List Currents toolbar for listing the currents vs. frequency at any segments on the selected wire. Also, if the wire has sources or loads, the lists of input impedances, voltages and powers are available.
This command is enabled when a wire is selected.

Plot Far-Field Pattern
This option has a sub-menu with the following commands:

3D Plot (F9)
Executes the AN-3D Pattern® program for plotting a three dimensional view of the radiation patterns.

2D Polar Plot (Ctrl+F9)
Displays the Radiation Pattern Cut dialog box for selecting a 2D cut of the 3D far-field pattern. Then, the selected 2D pattern cut is plotted in polar coordinates by the AN-Polar® program.

2D Rectangular Plot
Displays the Radiation Pattern Cut dialog box for selecting a 2D cut of the 3D far-field pattern. Then, the selected 2D pattern cut is plotted in rectangular coordinates by the AN-XY Chart® program.

Plot Far-Field Spectrum
Displays the Select Far-Field Point dialog box for selecting a particular point where the far-field components will be shown versus frequency. Then, this far-field spectrum is plotted in rectangular coordinates by the AN-XY Chart® program.

List Far-Field…
Displays the Select Far-Field Point dialog box for selecting a particular point where the far-field components will be shown versus frequency. Then, this far-field spectrum is listed in a table with different columns for the E-theta, E-phi, right and left polarized components.

Power Budget/RCS...
Displays the Power Budget dialog box for listing the total input power, consumed and radiated powers, power densities, efficiency, directivity and gain vs. frequency. In the case of plane wave excitation, the Radar Cross Section (RCS) vs. frequency is displayed.

Plot Near E-Field Pattern
This option has a sub-menu with the following commands:

> **3D Plot (F10)**
> Executes the AN-3D Pattern® program for plotting a three dimensional view of the near electric field components.
>
> **2D Plot (Ctrl+F10)**
> Displays the Near-Field Cut dialog box for selecting a 2D cut of the near electric field pattern. Then, the selected 2D pattern cut is plotted by the AN-XY Chart® program.

Plot Near E-Field Spectrum
Displays the Select Near-Field Point dialog box for selecting a point where the near electric field components will be shown versus frequency. Then, this near-field spectrum is plotted in rectangular coordinates by the AN-XY Chart® program.

List Near E-Field…
Displays the Select Near-Field Point dialog box for selecting a point where the near electric field components will be shown versus frequency. Then, this near-field spectrum is listed in a table with different columns for the field components.

Plot Near H-Field Pattern
This option has a sub-menu with the following commands:

> **3D Plot (F11)**
> Executes the AN-3D Pattern® program for plotting a three dimensional view of the near magnetic field components.
>
> **2D Plot (Ctrl+F11)**
> Displays the Near-Field Cut dialog box for selecting a 2D cut of the near magnetic field pattern. Then, the selected 2D pattern cut is plotted by the AN-XY Chart® program.

Plot Near H-Field Spectrum
Displays the Select Near-Field Point dialog box for selecting a point where the near magnetic field components will be shown versus frequency. Then, this near-field spectrum is plotted in rectangular coordinates by the AN-XY Chart® program.

List Near H-Field…
Displays the Select Near-Field Point dialog box for selecting a point where the near magnetic field components will be shown versus frequency. Then, this near-field spectrum is listed in a table with different columns for the field components.

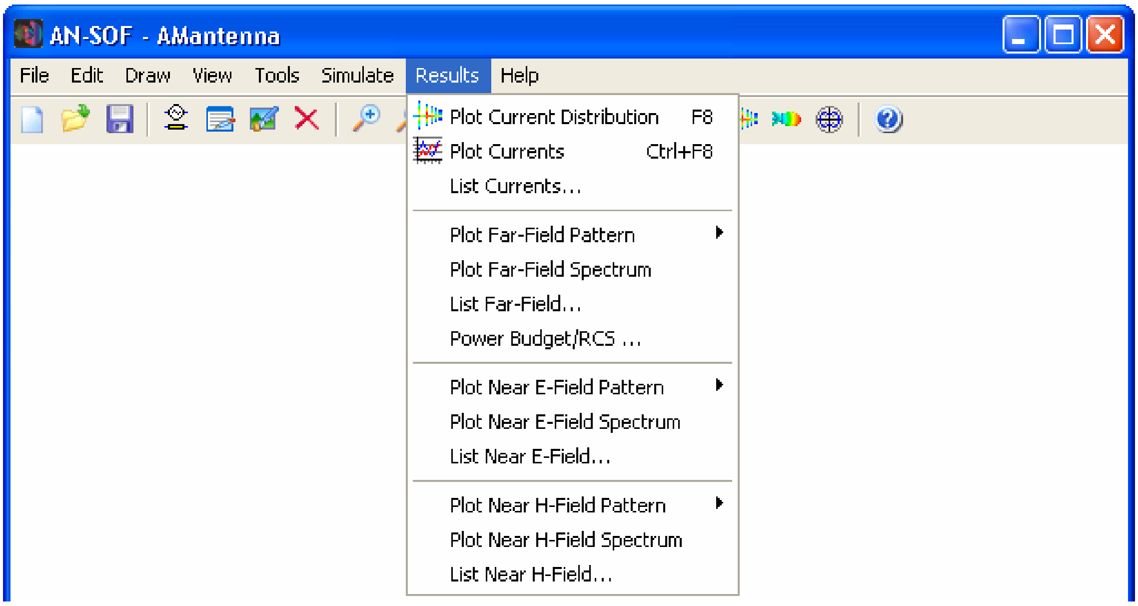


Fig. 3.8: Results menu.

Help Menu

Use the Help menu to access the AN-SOF® Help system, which is displayed in a special Help window. This menu has the following commands:

Contents
Displays the Help contents screen from which you can browse through topics by category.

Index
Displays the Help index screen from which you can type the word you are looking for.

AN-SOF® Home Page
Goes to the AN-SOF® web page: www.antennasoftware.com.ar in the default web browser.

Email Support Center
Executes the default e-mail software for sending a request for support to info@antennasoftware.com.ar.

About AN-SOF®
Shows copyright and version information for AN-SOF®.

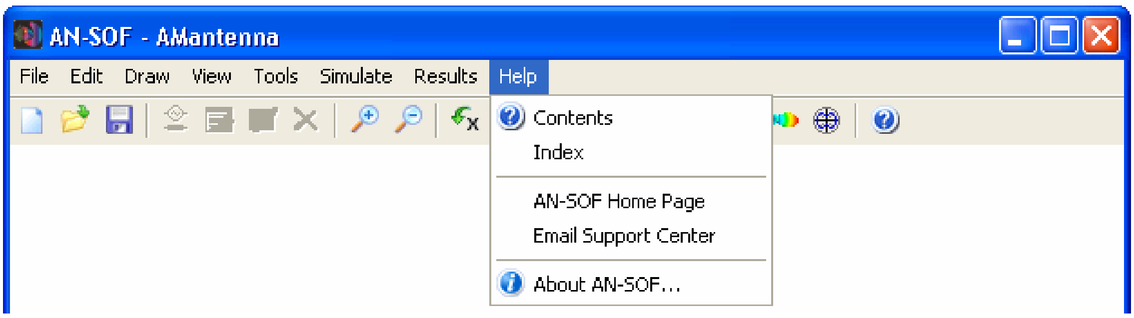


Fig. 3.9: Help menu.

3.2 Edit Toolbar

The Edit Toolbar has the following icons and associated commands:

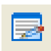
Fig. 3.10: Edit Toolbar.

New (Ctrl+N)
Creates a new project.

Open (Ctrl+O)
Displays the Open dialog box for loading an existing project (.EMM file).

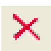
Save (Ctrl+S)
Saves the currently active project using its current name.

Source/Load toolbar (Ins)
Displays the Source/Load toolbar for exciting or loading the selected wire. This command is enabled when a wire is selected.



Modify (Ctrl+Ins)
Displays the Modify dialog box for modifying the selected wire.
This command is enabled when a wire is selected.

Wire color
Displays a Windows® dialog box for changing the color of the selected wire or wire grid. This command is enabled when a wire or wire grid is selected.



Delete (Del)
Deletes the selected wire, wire grid or group of wires with all sources and loads placed on it. This command is enabled when a wire, wire grid or group of wires is selected.

Zoom In (+)
Increases the size of the wire structure on the workspace.

Zoom Out (-)
Decreases the size of the wire structure on the workspace.

Rotate X
Enables the wire structure rotation around the x-axis.

Rotate Y
Enables the wire structure rotation around the y-axis.

Rotate Z
Enables the wire structure rotation around the z-axis.

Rotate
Performs a right-handed rotation of the wire structure around the selected axis when the upper arrow is pressed, and a left-handed rotation when the lower arrow is pressed.

Configure (Ctrl+C)
Displays the Configuration dialog box for defining the type of simulation, the operating frequencies of the system and other several options.

Run ALL (F5)
Runs the computation of the current distribution, far- and near-fields.

Plot Current Distribution (F8)
Executes the AN-3D Pattern® program for plotting the current distribution as a colored pattern over the wire structure.

Far-Field 3D Plot (F9)
Executes the AN-3D Pattern® program for plotting a three dimensional view of the radiation patterns.

Far-Field Polar Plot (Ctrl+F9)
Displays the Radiation Pattern Cut dialog box for selecting a 2D cut of the 3D far-field pattern. Then, the selected 2D pattern cut is plotted in polar coordinates by the AN-Polar® program.

Help
Displays the Help contents screen from which you can browse through topics by category.

3.3 Setting Up Preferences

General preferences include the unit system to be used for showing input and output data, the workspace appearance and the option to enable warning messages. AN-SOF® preferences can be accessed via Edit/Preferences from the main menu.

A suitable unit for frequencies, lengths, wire radius, inductances and capacitances can be selected in the Units page of the Preferences dialog box, Fig. 3.11. In the cases of lengths and radius, inches (in) and feet (ft) can be chosen apart from the standard SI units.

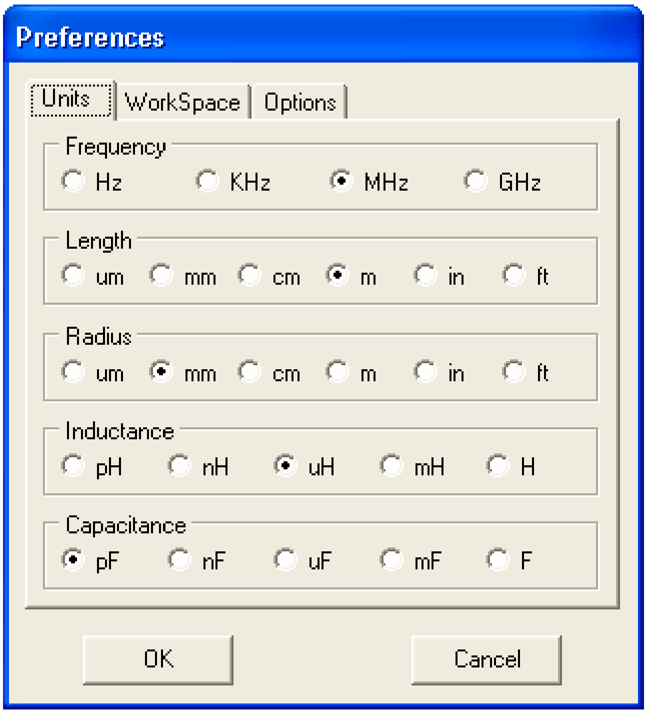


Fig. 3.11: Preferences dialog box. The Units tab is chosen, where the units for frequencies, lengths, wire radius, inductances and capacitances can be defined.

The workspace background color can be switched between black and white in the WorkSpace page tab, Fig. 3.12. Also, there are three levels for the pen width used to draw objects on the workspace: Thin, Medium and Thick. The pen width option applies to axes, wires, and wire grids.

In the Options page, check the Show Edit Toolbar option to see this toolbar. Besides, three warning questions can be set to avoid mistakes.

All of the preferences can be configured at any time, either before, during or after performing a simulation. When changing the unit systems, all of the input and output quantities will be rescaled automatically.

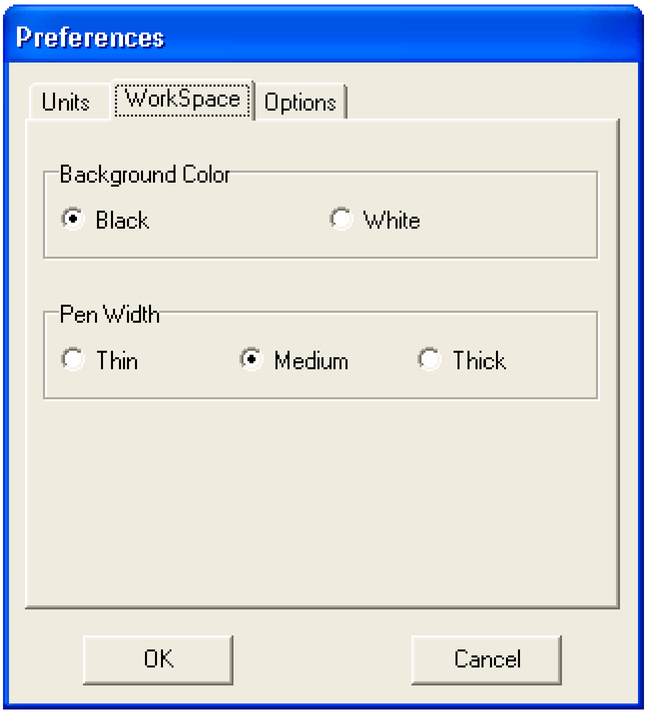


Fig. 3.12: Preferences dialog box. The WorkSpace tab is chosen, where the workspace background color and pen width can be selected.

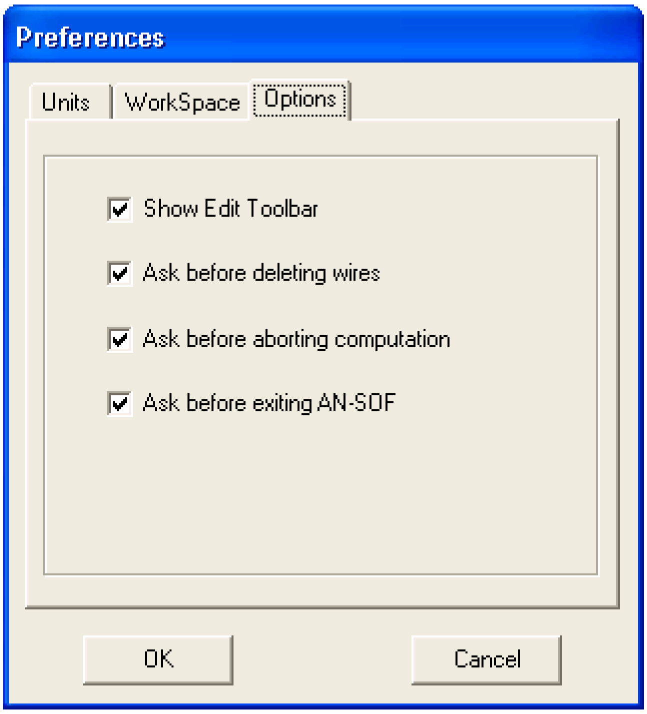


Fig. 3.13: Preferences dialog box. Use the Options page for setting up the confirmation or warning questions and showing the Edit Toolbar.

4. Configuring the Simulation

Choosing Simulate/Configure... in the main menu can display the Configuration dialog box, where the simulation can be configured. This dialog box has the following pages: Frequency, Medium, Far-Field, Near-Field, Incident Wave, Ground and Options, Fig. 4.1.

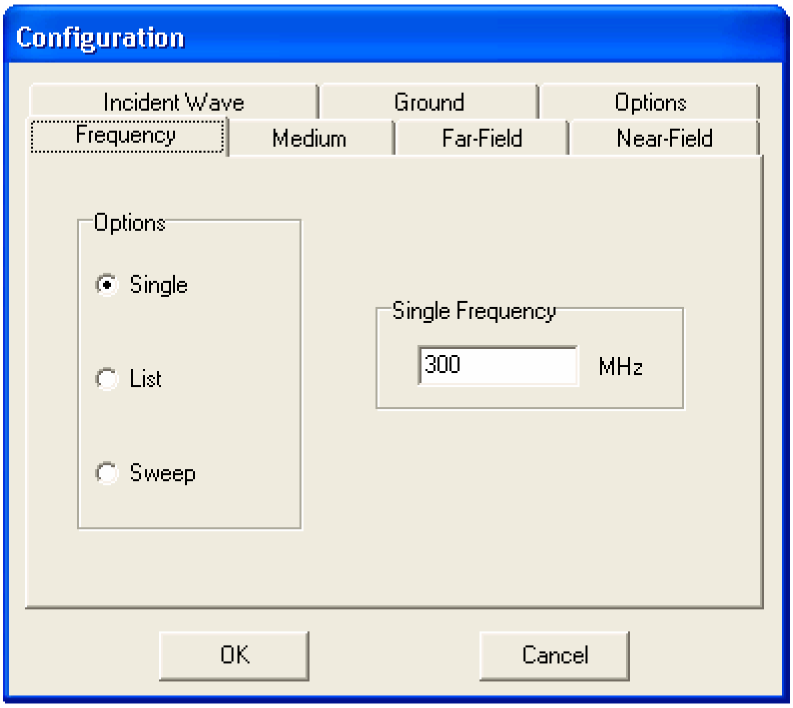


Fig. 4.1: Frequency page in the Configuration dialog box. A single frequency is entered.

- The **Frequency** page specifies the project operating frequencies.
- The **Medium** page sets the relative permittivity and permeability of the surrounding medium.
- The **Far-Field** page sets the angular ranges for the far-field computation.
- The **Near-Field** page sets the evaluation points in space for the near-fields computation.
- The **Incident Wave** page sets the incoming direction and polarization for the incident wave excitation.
- The **Ground** page defines the type of ground plane (PEC or real).
- The **Options** page specifies additional parameters, such as the reference impedance for VSWR and the accuracy of the simulation.

4.1 Defining the Frequencies

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Frequency page.

The Frequency page has three options: **Single**, **List** and **Sweep**. By choosing one of these options the computation can either be performed for a single frequency, for frequencies taken from a list or for a frequency sweep.

- If **Single** is chosen, write the frequency in the “Single Frequency” box, as shown in Fig. 4.1.

- If **List** is chosen, write the list of frequencies in the “Frequency List” box, Fig. 4.2. A list from a text file can be readed by pressing the Open button. The frequency list can also be saved to a text file by pressing the Save button.

- If **Sweep** is selected, it can either be linear or logarithmic. For a linear sweep the start, step and stop frequencies have to be defined. For a logarithmic frequency sweep the start, stop and a multiplication factor must be defined, Fig. 4.3.

The frequency unit can be changed going to Edit/Preferences in the main menu and choosing a suitable unit in the Units page of the Preferences dialog box.

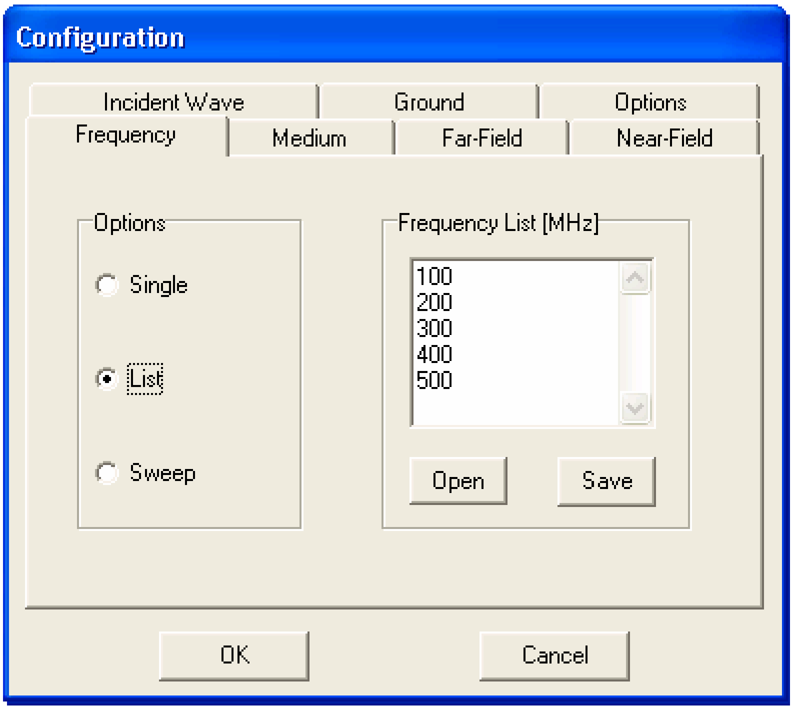


Fig. 4.2: Frequency page in the Configuration dialog box. A list of frequencies is entered.

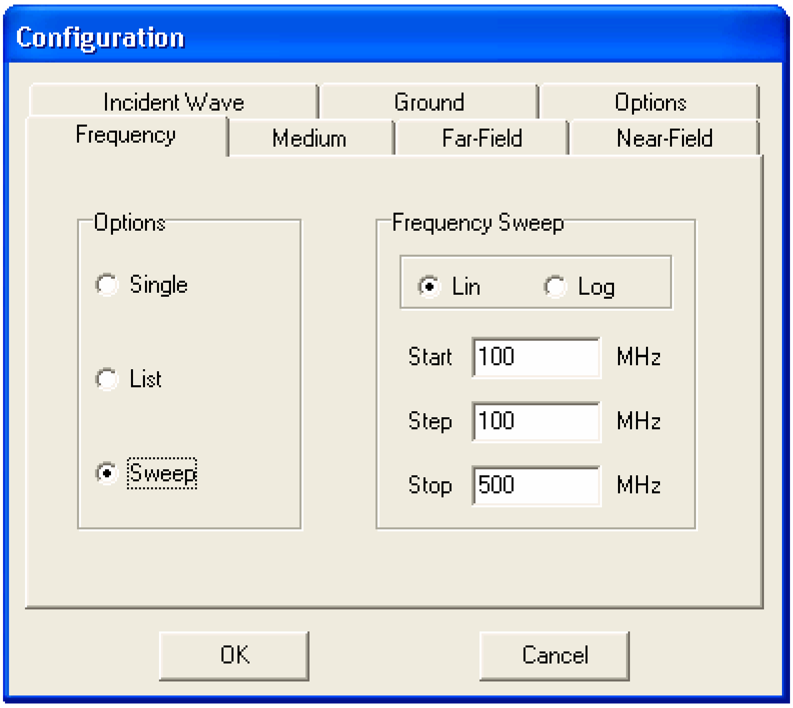


Fig. 4.3: Frequency page in the Configuration dialog box. A linear frequency sweep is entered.

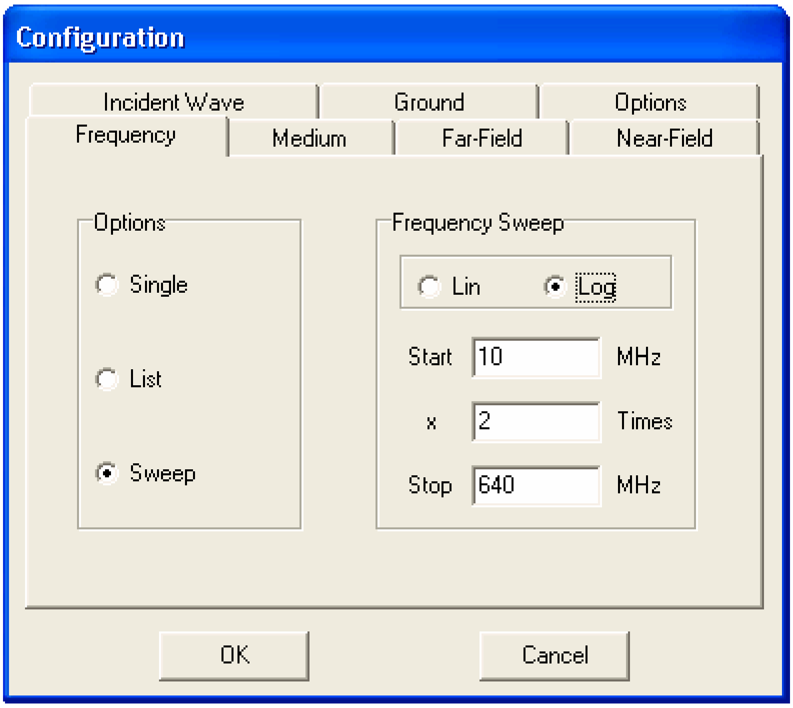


Fig. 4.4: Frequency page in the Configuration dialog box. A logarithmic frequency sweep is entered.

4.2 Defining the Environment

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Medium page.

The relative permittivity and permeability of the surrounding medium can be defined in the Medium page, Fig 4.5. The environment is assumed to be free space by default.

Two sets of values can be defined:

1. The relative permittivity and permeability of the medium for the computation of currents.
2. The relative permittivity and permeability of the medium for the computation of far-fields.

The first set of values will be used in the calculation of the current distribution when the space around wires is made of a material having different permittivity and permeability than free space. The radiated far-field can be computed in free space using the second set of values, but neglecting the effect of the material around wires. This can be used for experimentation and in most cases the far-field is computed in the same medium as the current distribution.

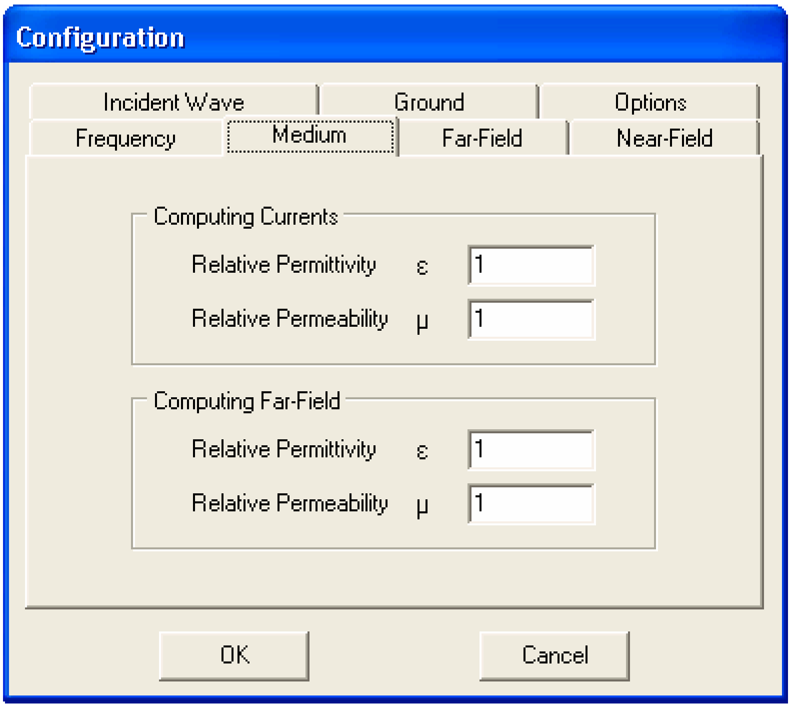


Fig. 4.5: Medium page in the Configuration dialog box.

4.3 Far-Fields

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Far-Field page, Fig. 4.6.

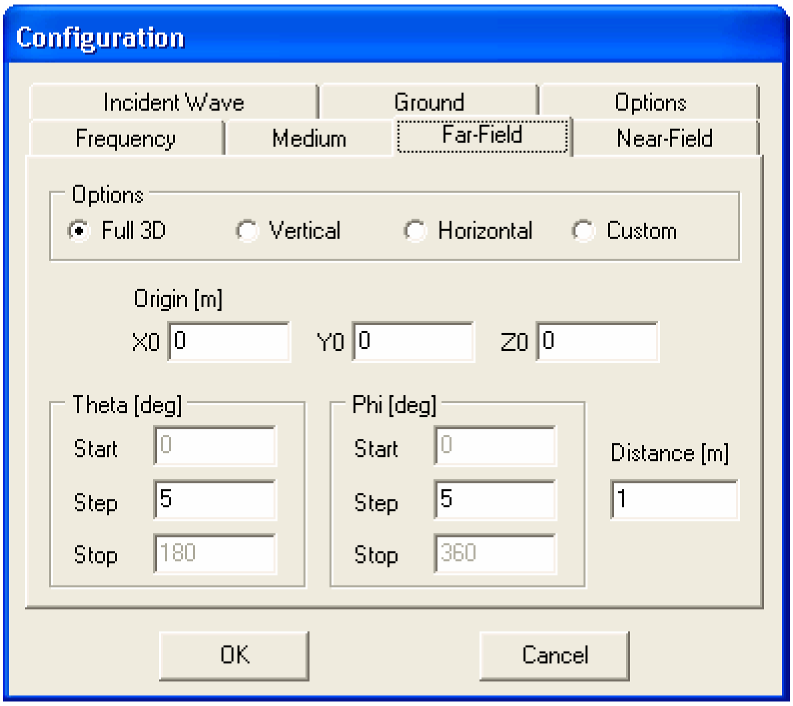


Fig. 4.6: Far-Field page in the Configuration dialog box.

The far-field can be computed once the current distribution has been obtained in a previous computation. Thus, the parameters defined in the Far-Field page have no effect in the determination of the currents and can be modified at any time.

There are four options for radiation pattern calculations:

Full 3D
The far-field is computed at directions covering the whole space to obtain a pattern having 3D radiation lobes. The steps for the Theta (zenith) and Phi (azimuth) angles can be entered in the **Theta [deg]** and **Phi [deg]** boxes.

Vertical
The far-field is computed at a vertical slice for a given Phi (azimuth) angle. The step for the Theta (zenith) angle can be entered in the **Theta [deg]** box, while the constant Phi can be defined in the **Phi [deg]** box.

Horizontal
The far-field is computed at a horizontal slice for a given Theta (zenith) angle. The step for the Phi (azimuth) angle can be entered in the **Phi [deg]** box, while the constant Theta can be defined in the **Theta [deg]** box.

Custom
The far-field is computed for the specified ranges of angles Theta (zenith) and Phi (azimuth). The start, step, and stop values for Theta and Phi can be entered in the **Theta [deg]** and **Phi [deg]** boxes.

Additionally, the following parameters can be defined:

Origin (X0,Y0,Z0)
This can be any point on the wire structure used as a phase reference, its coordinates do not affect the shape of the radiation pattern.

Distance
It is the distance from (X0,Y0,Z0) to an observation point in the far-field region. A normalized far-field pattern can be obtained by defining Distance = 1.

The zenith and azimuth angles, θ (Theta) and ϕ (Phi), are shown in Fig. 4.7, where it is also shown de Distance R from the structure to an observation point in the far-field zone. These three numbers (R,θ,ϕ) define the spherical coordinates of the far-field point.

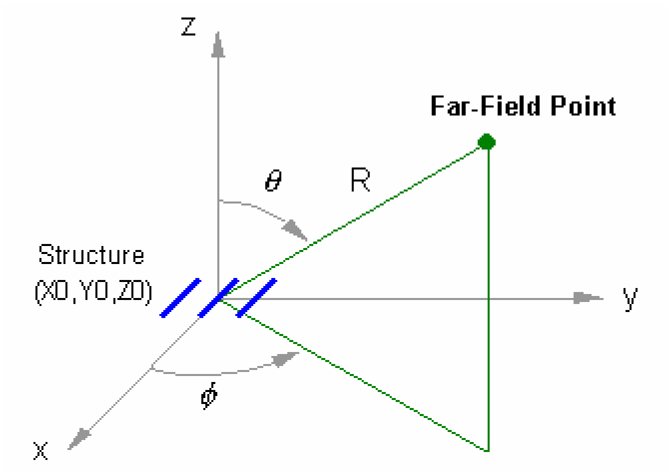


Fig. 4.7: Spherical coordinates (R,θ,ϕ) of a far-field point.

Important Information

In order to check the average radiated power of a structure and compute the Radar Cross Section (RCS) in the case of plane wave excitation, a full radiation pattern covering the whole of space should be defined. For this reason, the Theta and Phi angles should vary in the following ranges when the **Custom** option is chosen:

If the environment is **free space** (there is no ground plane):

$0 \leq$ Theta ≤ 180 [degrees]

and

$0 \leq$ Phi ≤ 360 [degrees]

If the environment has a **ground plane**:

$0 \leq$ Theta ≤ 90 [degrees]

and

$0 \leq$ Phi ≤ 360 [degrees]

These angular ranges allow the Average Power Density to be computed averaging the power density or Poynting vector in all directions in 3D space. If there is a ground plane, directions must be considered in half-space.

4.4 Near-Fields

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Near-Field page, Fig. 4.8.

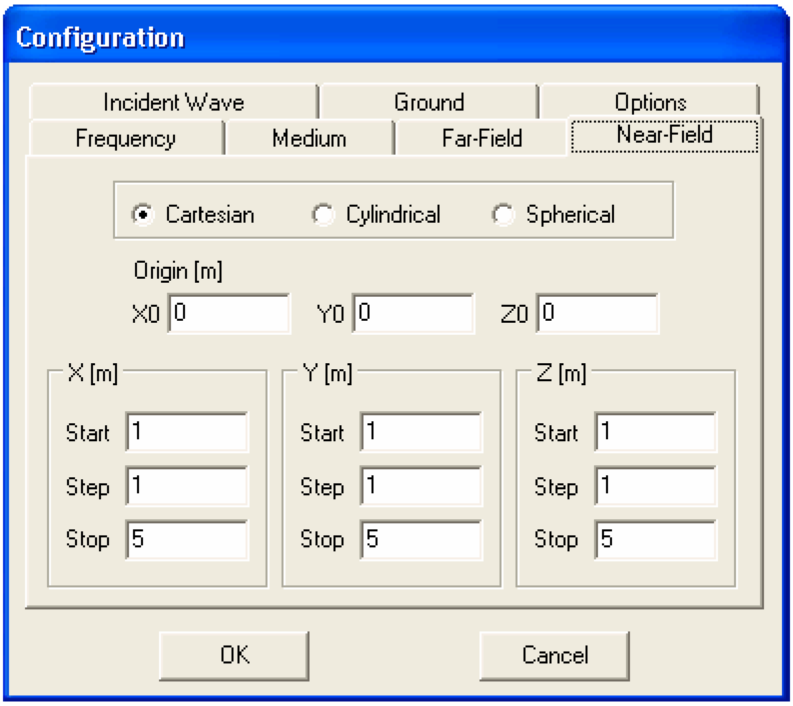


Fig. 4.8: Near-Field page in the Configuration dialog box. The Cartesian option is selected.

The near-field can be computed once the current distribution has been obtained in a previous computation. Thus, the parameters defined in the Near-Field page have no effect in the determination of the currents, and can be set at any time.

The Near-Field page has three options: **Cartesian**, **Cylindrical**, and **Spherical**. By choosing one of these options near-fields can either be computed in Cartesian, Cylindrical or Spherical coordinates.

If the Cartesian option is chosen, the following parameters can be defined for near-field calculations, Fig. 4.8:

Origin (X0,Y0,Z0)
This point is the origin of the Cartesian coordinates used to define the points in space where near-fields will be calculated.

X
This box is used to define x-coordinates of the points in space where near-fields will be calculated. The start, step and stop x-coordinates have to be defined. Start and stop x-coordinates are measured from X0.

Y
This box is used to define y-coordinates of the points in space where near-fields will be calculated. The start, step and stop y-coordinates have to be defined. Start and stop y-coordinates are measured from Y0.

Z
This box is used to define z-coordinates of the points in space where near-fields will be calculated. The start, step and stop z-coordinates have to be defined. Start and stop z-coordinates are measured from Z0.

If the Cylindrical option is chosen, the following parameters can be defined for near-field calculations, Fig. 4.9:

Origin (X0,Y0,Z0)
This point is the origin of the Cylindrical coordinates used to define the points in space where near-fields will be calculated.

R
This box is used to define the distances or R-coordinates of the points in space where near-fields will be calculated. The start, step and stop R-coordinates have to be defined. Start and stop distances or R-coordinates are measured from the origin point (X0,Y0,Z0).

Phi [deg]
This box is used to define the azimuth angles or phi-coordinates of the points in space where near-fields will be calculated. The start, step and stop theta-coordinates have to be defined in degrees.

Z
This box is used to define the z-coordinates of the points in space where near-fields will be calculated. The start, step and stop z-coordinates have to be defined.

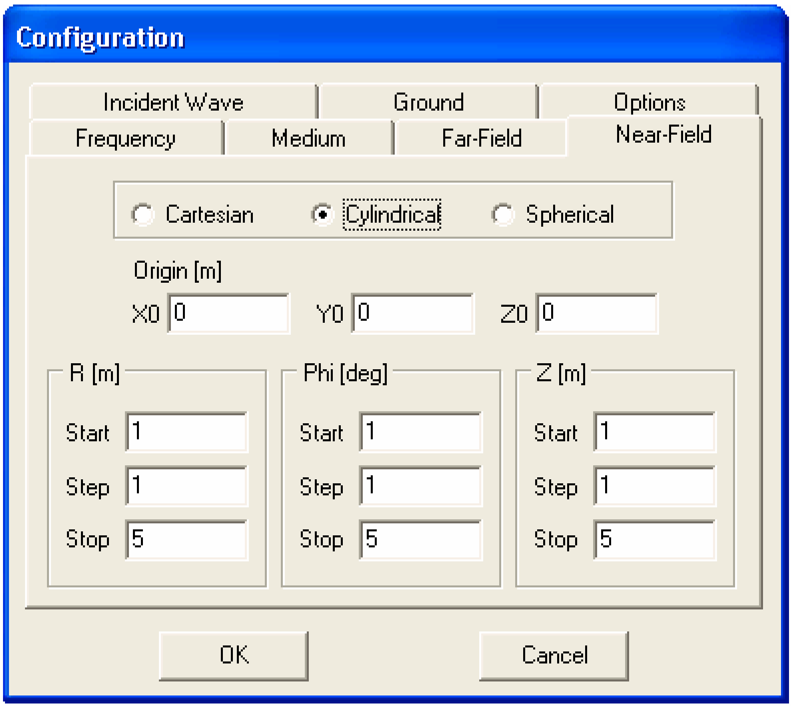


Fig. 4.9: Near-Field page in the Configuration dialog box. The Cylindrical option is selected.

If the Spherical option is chosen, the following parameters have to be defined for near-field calculations, Fig. 4.10:

Origin (X0,Y0,Z0)
This point is the origin of the Spherical coordinates used to define the points in space where near-fields will be calculated.

R
This box is used to define the distances or R-coordinates of the points in space where near-fields will be calculated. The start, step and stop R-coordinates have to be defined. Start and stop distances or R-coordinates are measured from the origin point (X0,Y0,Z0).

Theta [deg]
This box is used to define zenith angles or theta-coordinates of the points in space where near-fields will be calculated. The start, step and stop theta-coordinates have to be defined in degrees.

Phi [deg]
This box is used to define azimuth angles or phi-coordinates of the points in space where near-fields will be calculated. The start, step and stop phi-coordinates have to be defined in degrees.

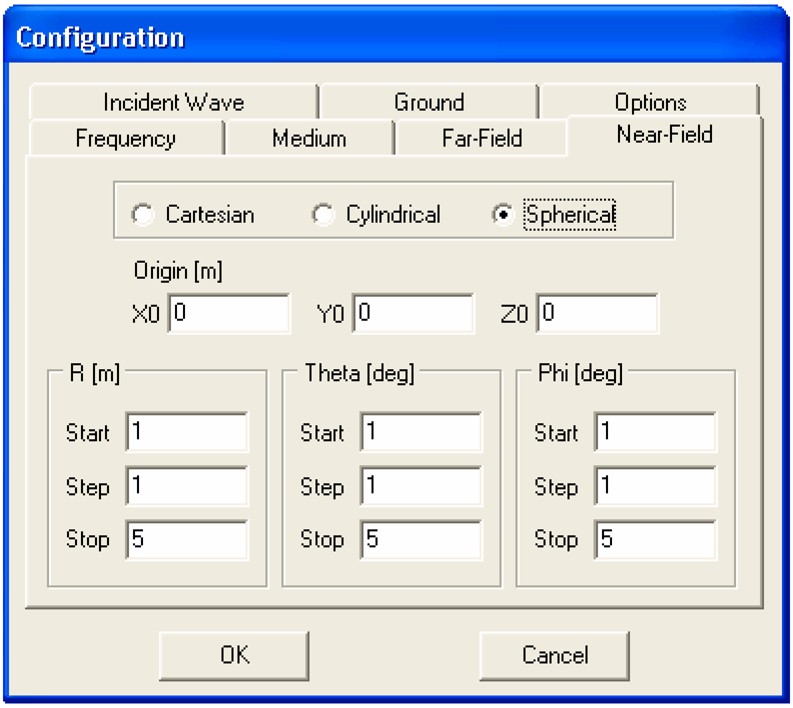


Fig. 4.10: Near-Field page in the Configuration dialog box. The Spherical option is selected.

4.5 Incident Wave

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Incident Wave page, Fig. 4.11.

The parameters set in this page are taken into account only if the option **Incident Plane Wave** in the Options page of the Configuration dialog box is checked.

When an incident plane wave is used as excitation, all discrete sources, if any, will not be considered in the simulation.

The following parameters have to be defined for the incident wave excitation:

E-Field Major Axis [V/m]
In the case of linear polarization, it is the amplitude, in Volts per meter (rms value), of the incoming electric field. For an elliptically polarized plane wave, it is the major axis of the polarization ellipse.

Axial Ratio
It is the ratio of the minor axis to the major axis of the polarization ellipse. If the axial ratio is positive a right-handed ellipse is obtained, and if the axial ratio is negative a left-handed ellipse is defined.
A linearly polarized wave can be defined if the axial ratio is set to zero.

Phase Reference [deg]
It is the phase, in degrees, of the incident plane wave at the origin of coordinates and can be used to change the phase reference in the calculation. Its value only shifts all phases in the structure by the same amount.

Gamma [deg]
For a linearly polarized wave, it is the polarization angle, in degrees, of the incident electric field measured from the plane of incidence to the direction of the electric field vector as it is shown in Fig. 4.12.
For an elliptically polarized wave, Gamma is the angle between the plane of incidence and the major ellipse axis.

Theta [deg]
It is the zenith angle, in degrees, of the incident direction.

Phi [deg]
It is the azimuth angle, in degrees, of the incident direction.

The definition of these parameters is illustrated in Fig. 4.12.

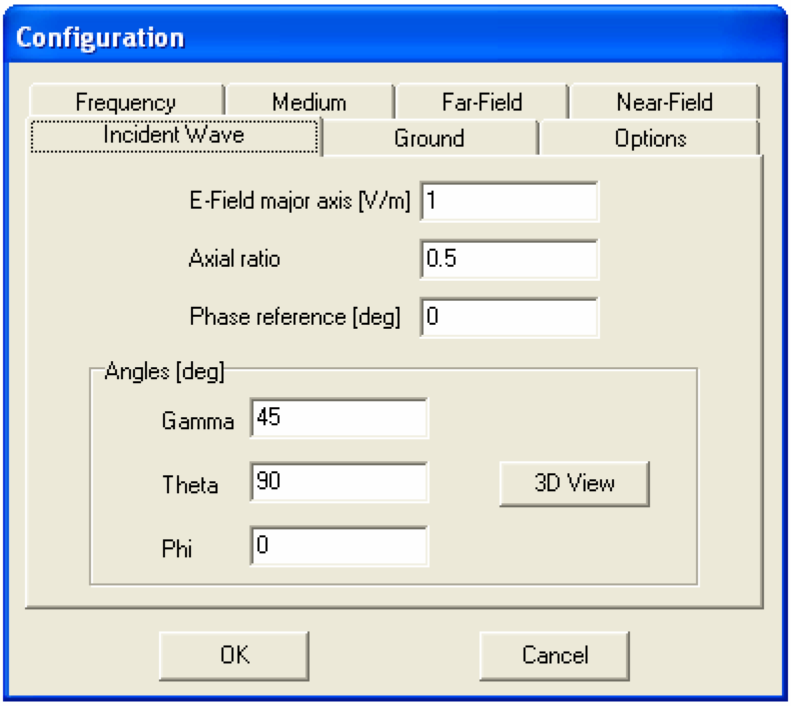


Fig. 4.11: Incident Wave page in the Configuration dialog box.

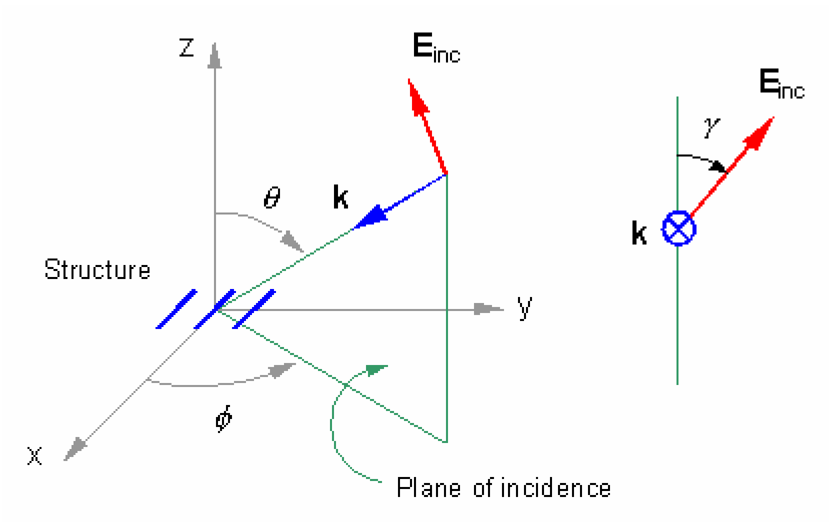


Fig. 4.12: Definition of the incident plane wave.

When the **3D View** button is pressed a special user interface is enabled. With this tool the direction of arrival of the plane wave and its polarization can be specified in an easy way, Fig. 4.13.

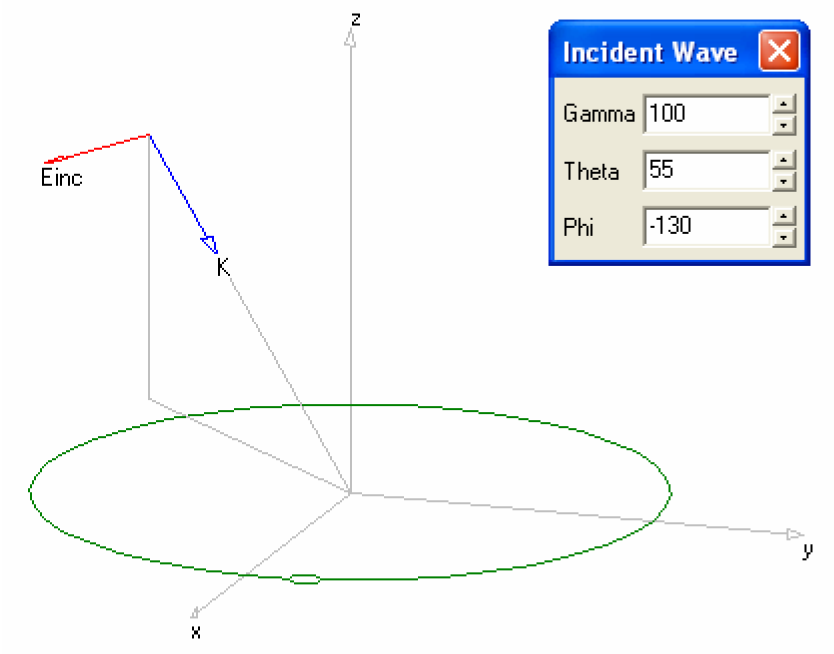


Fig. 4.13: 3D View user interface for the incident wave definition. In the case of elliptical polarization, the electric field vector Einc indicates the major ellipse axis.

4.6 Ground Plane

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Ground page, Fig. 4.14. The following options can be selected:

None
None ground plane is used. The simulation is performed in free space with the permittivities and permeabilities defined in the Medium page of the Configuration dialog box.

Perfect
An infinite perfectly electric conducting (PEC) ground plane will be placed at the specified height from the xy-plane. Thus, the ground plane is parallel to the xy-plane. The Height value defines the ground plane height above the xy-plane (a negative Height defines the ground plane below the xy-plane).

Real
A real ground plane with the permittivity, permeability and conductivity defined by the user is placed on the xy-plane. The real ground is only used to compute the near- and far-fields radiated from the structure using the Sommerfeld-Norton approximation and the Fresnel's reflection coefficients, respectively. The current flowing on the wire structure is calculated using a PEC ground plane.

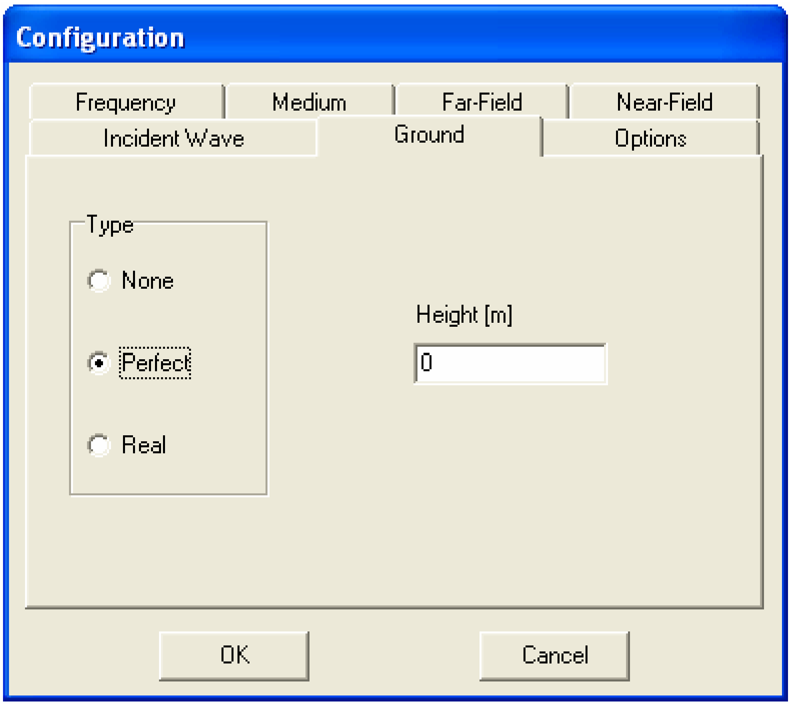


Fig. 4.14: Ground page in the Configuration dialog box. The Perfect ground option is selected.

4.7 Options

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Options page. Four options for the type of simulation are available in the Options page, Fig. 4.15.

If **Incident Plane Wave** is checked, an incident plane wave will be used as the excitation of the wire structure. All discrete sources on the wires will not be used in the simulation. The direction of incidence and polarization of the incoming field can be set in the Incident Wave page of the Configuration dialog box, which is described in Section 4.5.

If **NGF** is checked, the Numerical Green's Function calculation is performed in the simulation, that is, the LU-decomposed matrix of the system is stored in a file in the first computation. Then, by using the stored information, new simulations are performed faster than the first one.

If **Load Impedances** is checked, lumped impedances will be taken into account in the simulation. With this option all of the lumped loads can be disabled or enabled at the same time.

If **Wire Resistivity** is checked, the finite resistivity of the wires will be taken into account in the simulation. Any wire has its own resistivity in [Ohm meter] that is defined when the wire is drawn. This option allows considering the whole structure as a perfect electric conductor when it is unchecked.

If **Wire Coating** is checked, the coating materials of the wires will be taken into account in the simulation. Any wire has its own coating specified by a dielectric permittivity, magnetic permeability and thickness, which are defined when the wire is drawn. When this option is unchecked, the wire coating is not considered in the simulation.

In the Options page, the **Reference Impedance for VSWR** calculations can be defined. A default value of 50 Ohm is selected, Fig 4.15.

The accuracy of the integrals involved in the calculations can be set in the Options page.

The **Tolerance** is the error in the evaluation of interactions between wire segments which are separated by a distance less than the **Interaction Distance**.

The **Interaction Distance** is the maximum distance (in wavelengths) between segments for which an error less than the Tolerance is guaranteed in the integrations. The interaction between all wire segments further apart than the Interaction Distance is computed using a third-degree polynomial approximation to the involved integrals, which is more accurate for curved segments than the Hertzian dipole approximation used in the standard MoM. Therefore, the Interaction Distance could be set to zero for a faster computation when wire

segments are not too close to each other, but results will be less accurate. A convergence test for various values of this parameter is recommended.

For most cases, a Tolerance between 0.1% and 1% and an Interaction Distance between 0.25 and 1.0 wavelengths are enough for accurate results.

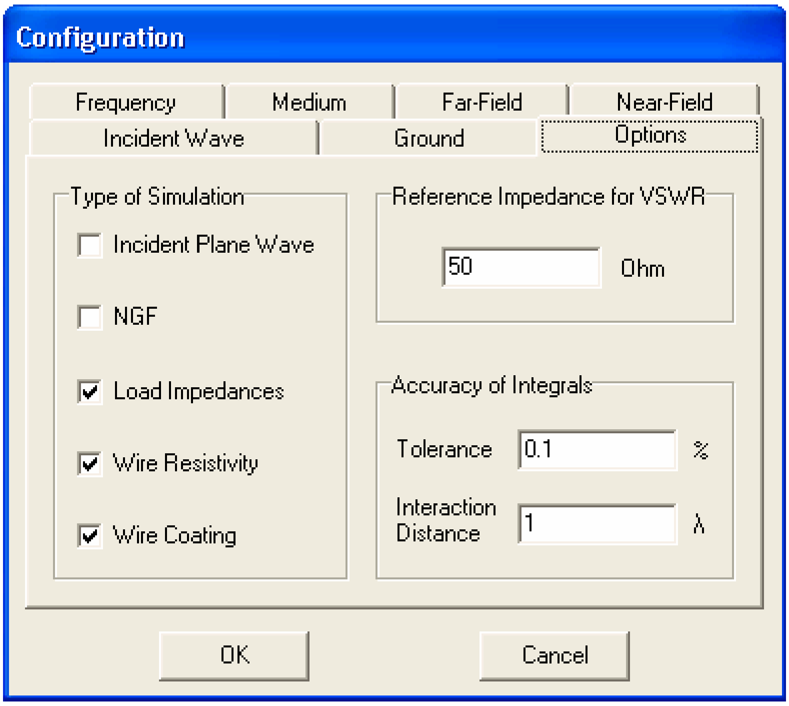


Fig. 4.15: Options page in the Configuration dialog box.

5. Drawing Wires

AN-SOF® has different types of wires. Any wire type has its own geometrical parameters, attributes and materials that can be set in its specific Draw dialog box. This dialog box allows creating and drawing a new wire on the project workspace.

Choosing Draw/Wire in the main menu shows a sub-menu with the following commands:

- **Line:** Displays the Draw dialog box for drawing a linear or straight wire.
- **Arc:** Displays the Draw dialog box for drawing an arc or arced wire.
- **Circle:** Displays the Draw dialog box for drawing a circular loop.
- **Helix:** Displays the Draw dialog box for drawing a helix or helical wire.
- **Quadratic:** Displays the Draw dialog box for drawing a quadratic wire.
- **Archimedean Spiral:** Displays the Draw dialog box for drawing an Archimedean spiral.
- **Logarithmic Spiral:** Displays the Draw dialog box for drawing a logarithmic spiral.

These commands can also be chosen from a pop-up menu by clicking the right mouse button on the workspace, as shown in Fig. 5.1.

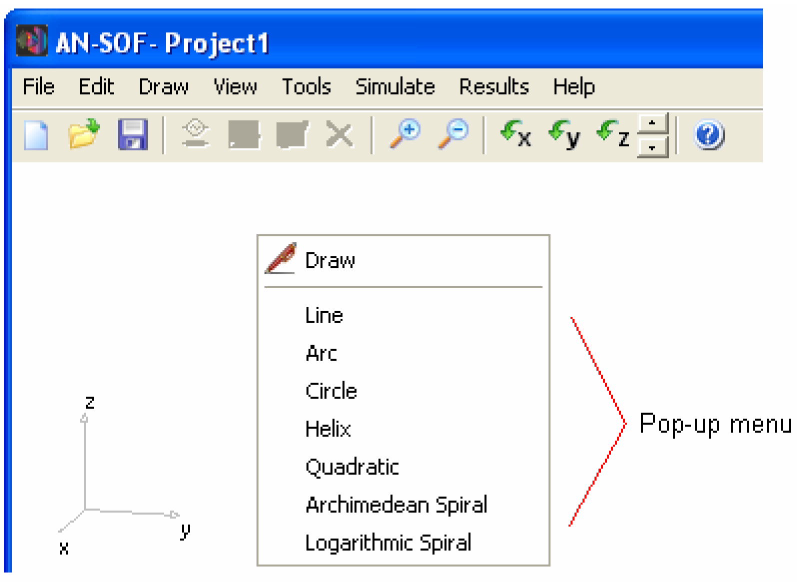


Fig. 5.1: Pop-up menu in the workspace.

5.1 Line

The Line refers to a linear or straight wire.

Choose Draw/Wire/Line in the main menu to display the Draw dialog box for the Line, Fig. 5.2. This dialog box has two pages: Line and Attributes, Fig. 5.3.

The Line page

The Line page sets the geometrical parameters for the Line.
In this page there are two options: 2 Points and Start - Direction - Length.

The **2 Points** option allows entering the Line by giving two points: "From Point" and "To Point", as shown in Figs. 5.3 and 5.4.

If **Start - Direction - Length** is chosen, the line will be drawn starting from Start Point, in the direction defined by the Theta and Phi angles in spherical coordinates, and ending at a point defined by the wire Length measured in that direction, Figs. 5.5 and 5.6.

Once the geometrical parameters in the Line page have been set, the Attributes page can be chosen. Section 5.8 describes the parameters that can be defined in the Attributes page. The wire resistivity and coating can be set in the Materials page described in Section 5.9.

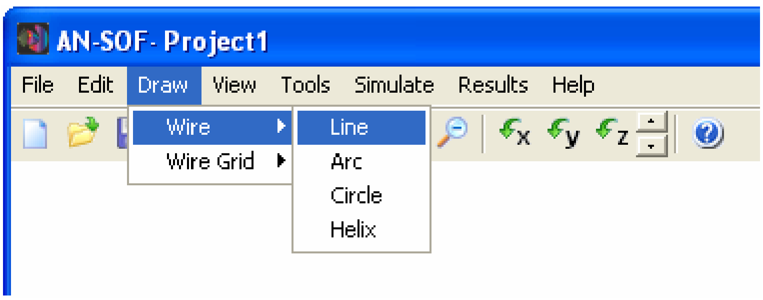


Fig. 5.2: The Draw/Wire/Line command in the main menu displays the Draw dialog box for the Line.

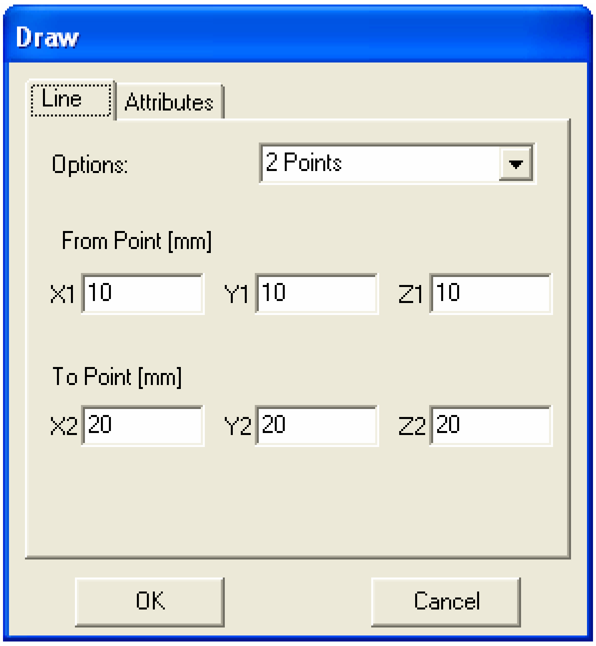


Fig. 5.3: "2 Points" option in the Line page of the Draw dialog box.

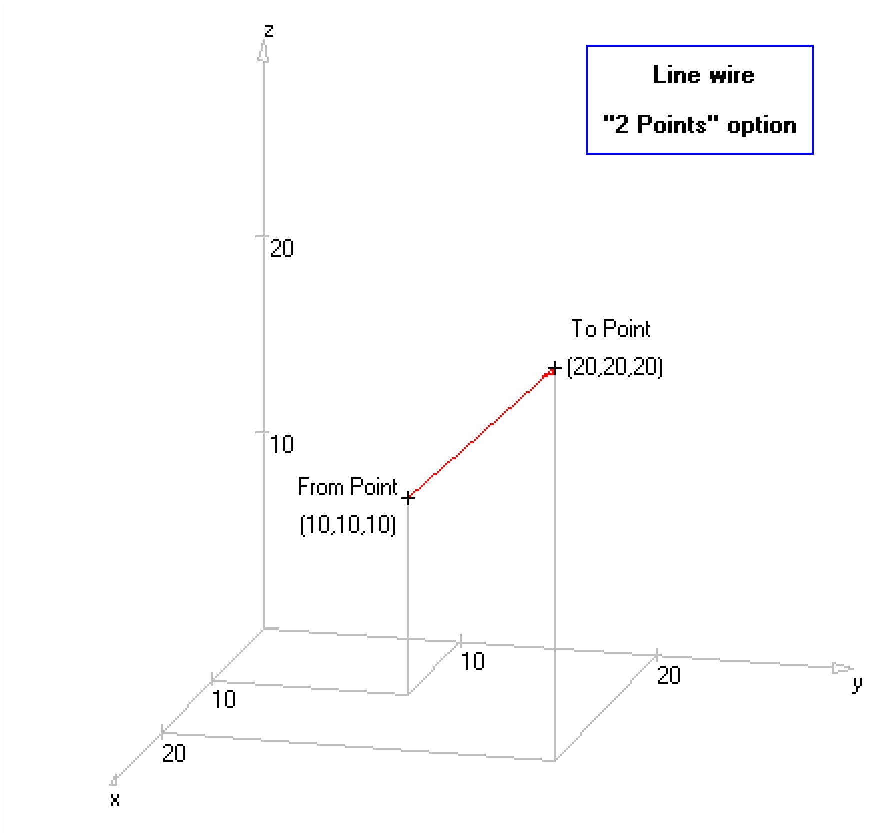


Fig. 5.4: A Line drawn using the "2 Points" option.

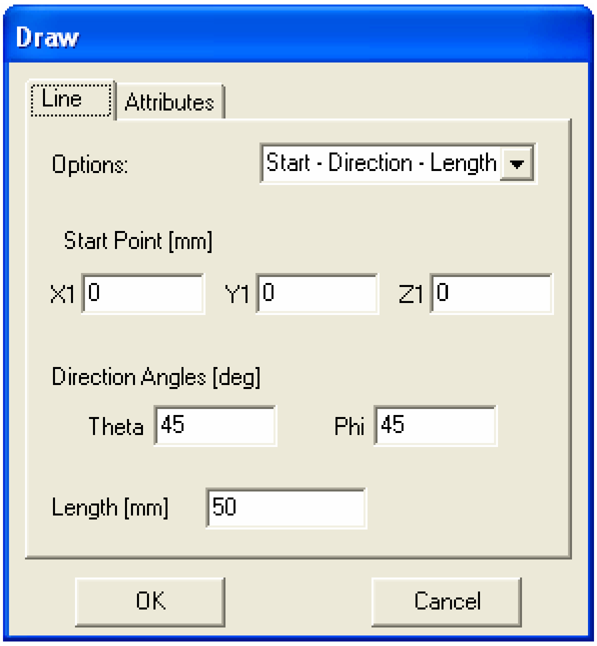


Fig. 5.5: "Start - Direction - Length" option in the Line page of the Draw dialog box.

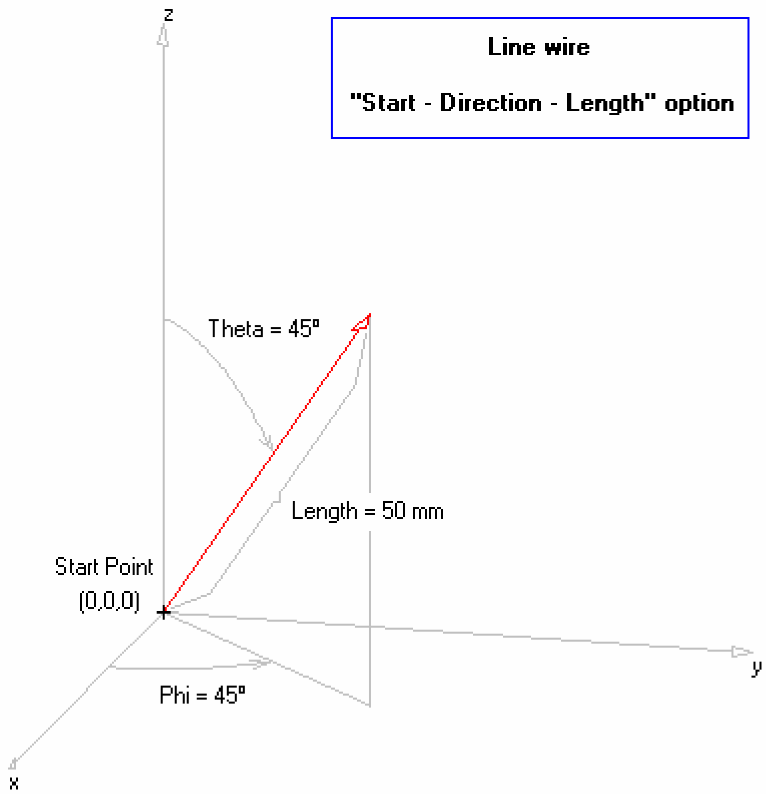


Fig. 5.6: A Line drawn using the "Start - Direction - Length" option.

5.2 Arc

The Arc refers to a circular arc or arced wire.

Choose Draw/Wire/Arc in the main menu to display the Draw dialog box for the Arc, Fig. 5.7. This dialog box has two pages: Arc and Attributes, Fig. 5.8.

The Arc page

The Arc page sets the geometrical parameters for the Arc.
In this page there are two options: 3 Points and Start - Center - End.

The **3 Points** option allows entering the Arc by giving three points. An arc starting from Start Point, passing through Second Point and ending at End Point will be drawn on the workspace, Figs. 5.8 and 5.9.

If **Start - Center - End** is chosen, the Arc will be drawn starting from Start Point, with the center defined by Center and ending at a point determined by End Point, Figs. 5.10 and 5.11. The End Point determines the arc aperture angle and the plane where it will be on, so this point could not coincide with the actual ending point of the arc.

Once the geometrical parameters in the Arc page have been set, the Attributes page can be chosen. Section 5.8 describes the parameters that can be defined in the Attributes page. The wire resistivity and coating can be set in the Materials page described in Section 5.9.

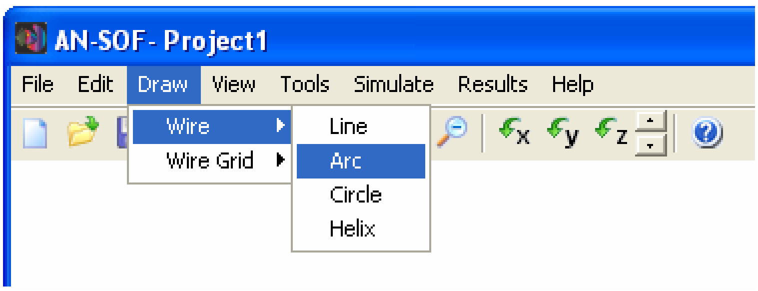


Fig. 5.7: The Draw/Wire/Arc command in the main menu displays the Draw dialog box for the Arc.

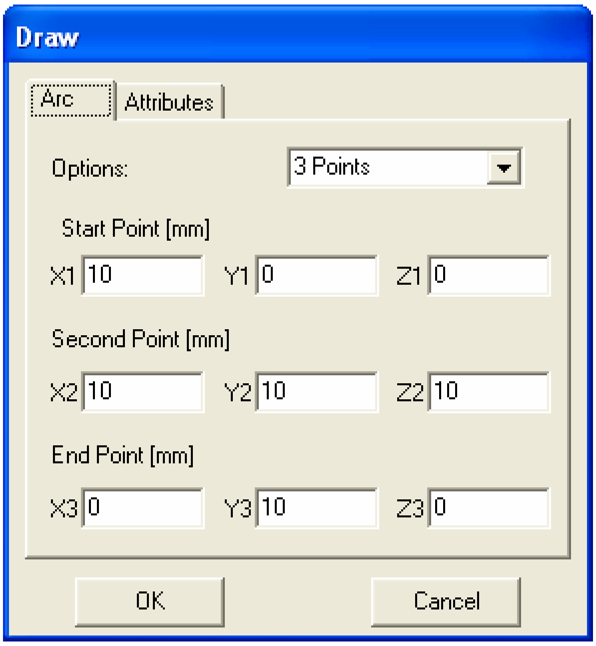


Fig. 5.8: "3 Points" option in the Arc page of the Draw dialog box.

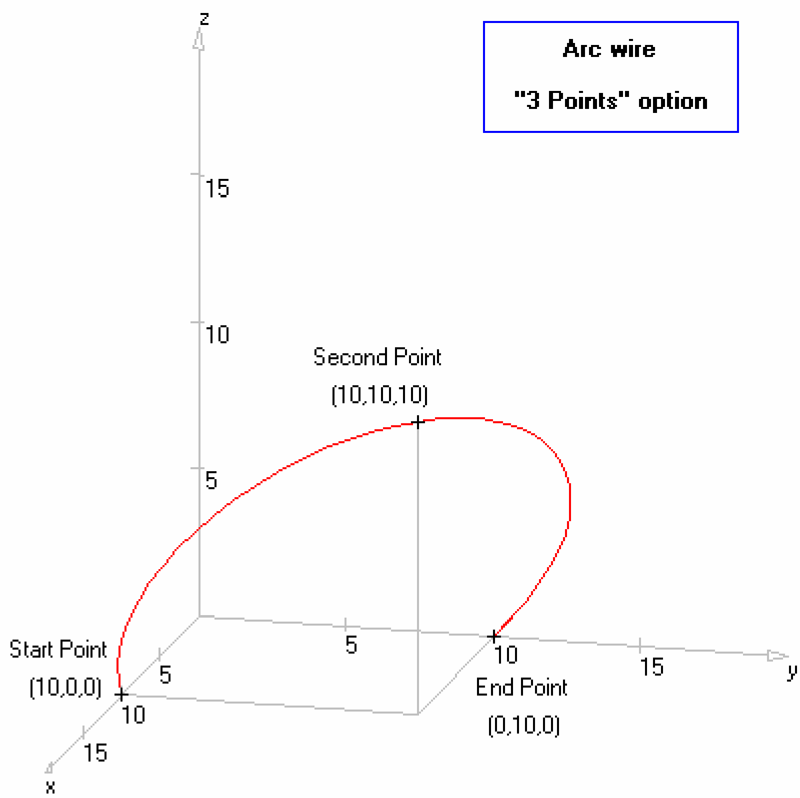


Fig. 5.9: An Arc drawn using the "3 Points" option.

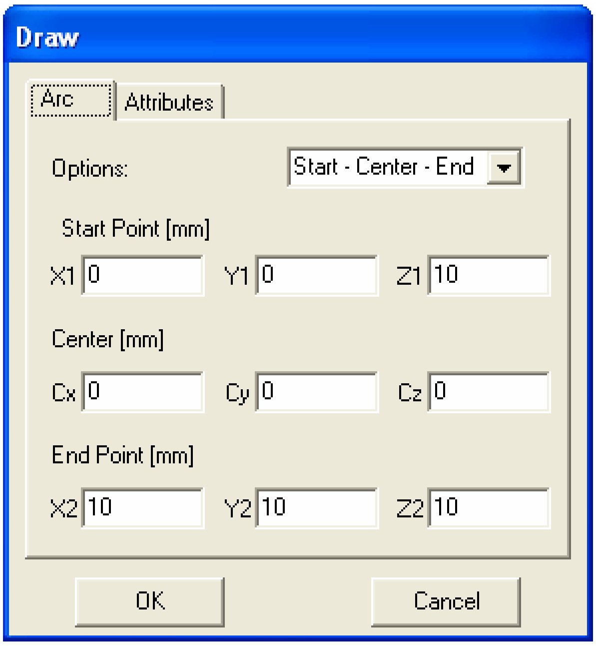


Fig. 5.10: "Start - Center - End" option in the Arc page of the Draw dialog box.

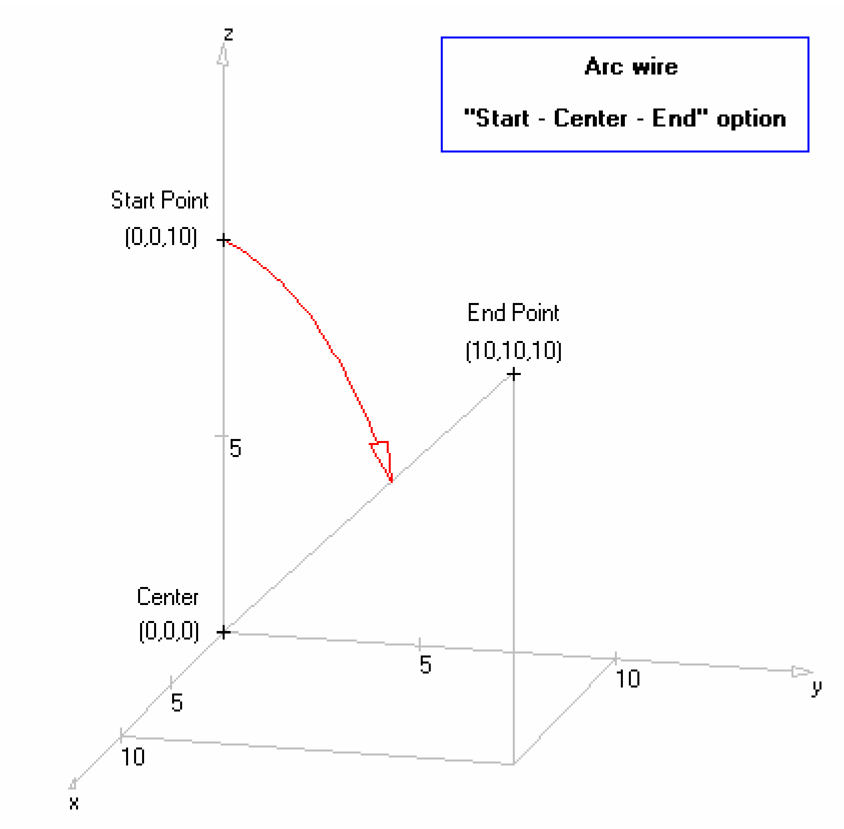


Fig. 5.11: An Arc drawn using the "Start - Center - End" option.

5.3 Circle

The Circle refers to a circular loop.
Choose Draw/Wire/Circle in the main menu to display the Draw dialog box for the Circle, Fig. 5.12. This dialog box has three pages: Circle, Orientation and Attributes, Fig. 5.13.

The Circle page

The Circle page sets the geometrical parameters for the Circle.
In this page there are two options: Center - Radius - Orientation and 3 Points.

The **Center - Radius - Orientation** option allows entering the Circle by giving its Center, Radius, and axis, Figs. 5.13 and 5.14. The circle axis can be set in the Orientation page.
If the **3 Points** option is chosen, the Circle will be drawn starting from First Point, passing through Second Point and Third Point, and ending at First Point, Figs. 5.15 and 5.16. Thus, the circle starts and ends at the same point. The Orientation page will be invisible when the 3 Points option is chosen.

Once the geometrical parameters in the Circle page, and eventually in the Orientation page, have been set the Attributes page can be chosen. Section 5.8 describes the parameters that can be defined in the Attributes page. The wire resistivity and coating can be set in the Materials page described in Section 5.9.

The Orientation page

The Orientation page sets the orientation for the Circle.
In this page there is a box with two options: Angles and Vector.

If **Angles** is selected, the circle axis can be defined by given an orthogonal direction to the rest plane of the circle. Thus, the Theta and Phi angles determine the axis direction in spherical coordinates, Fig. 5.17.
If **Vector** is selected, the circle axis can be defined by given an orthogonal vector to the rest plane of the circle. Thus, the Nx, Ny, and Nz components of that vector determine the axis direction, Fig. 5.18.
The circle can be rotated around its axis by given the Rotation angle, Figs 5.17 and 5.18.

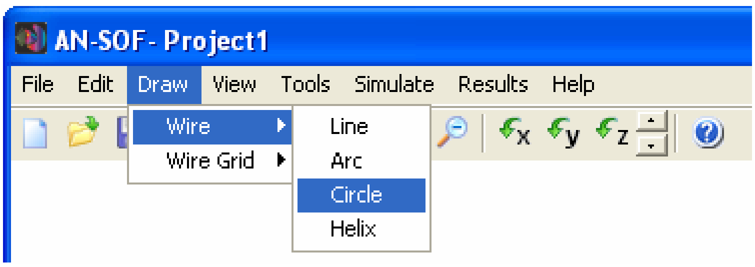


Fig. 5.12: The Draw/Wire/Circle command in the main menu displays the Draw dialog box for the Circle.

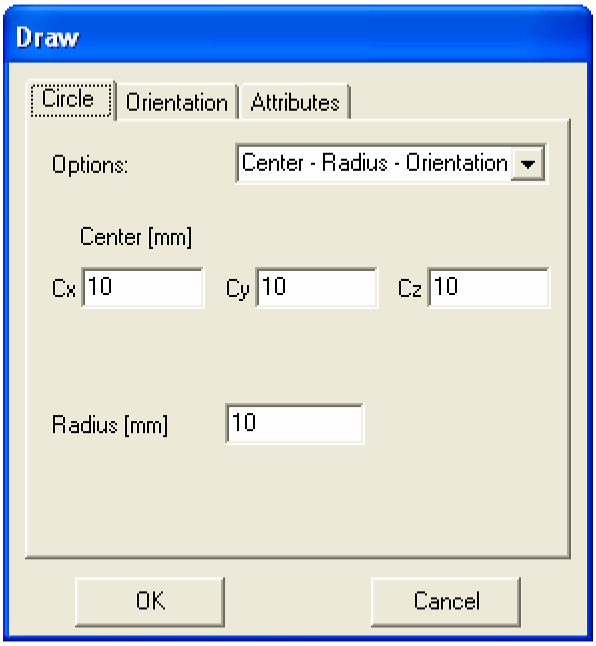


Fig. 5.13: "Center - Radius - Orientation" option in the Circle page of the Draw dialog box.

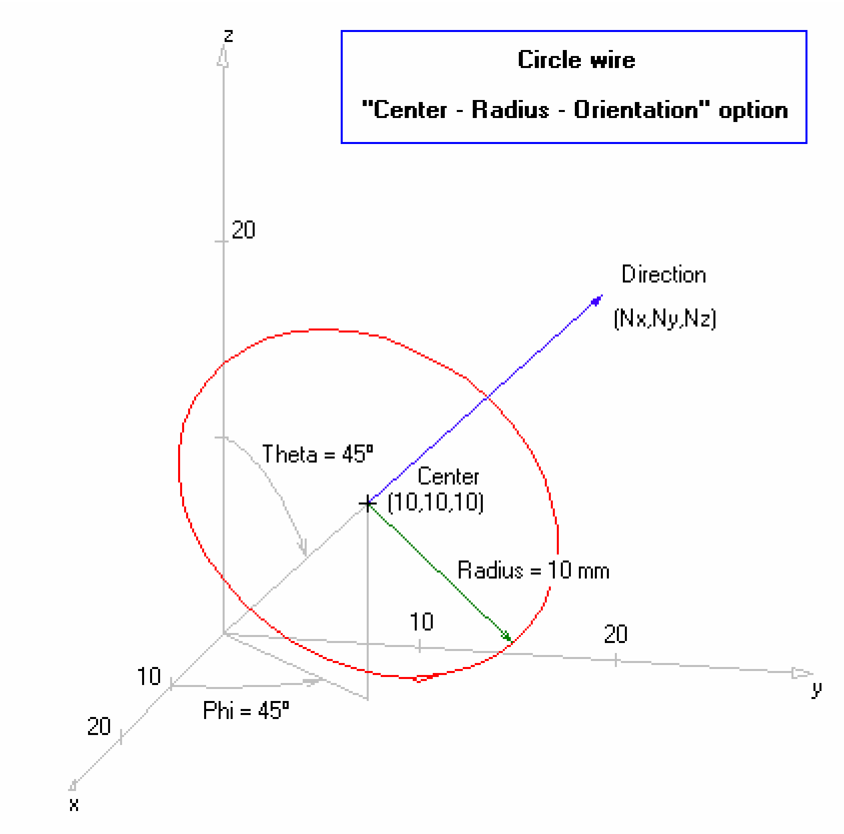


Fig. 5.14: A Circle drawn using the "Center - Radius - Orientation" option.

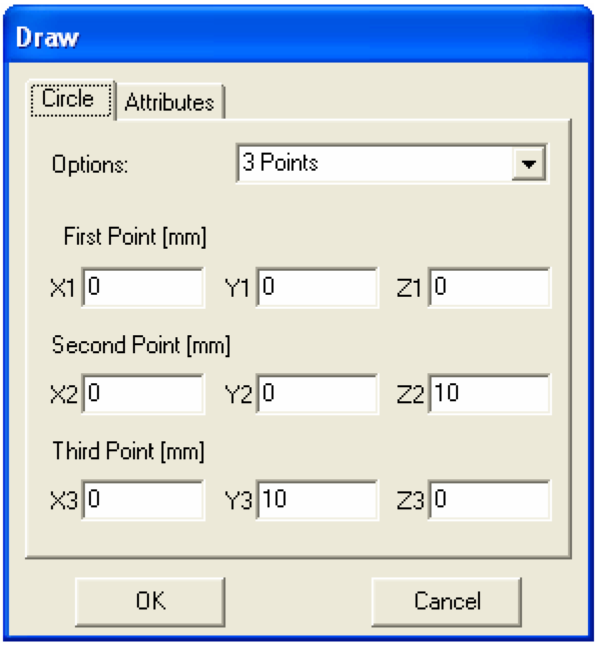


Fig. 5.15: "3 Points" option in the Circle page of the Draw dialog box.

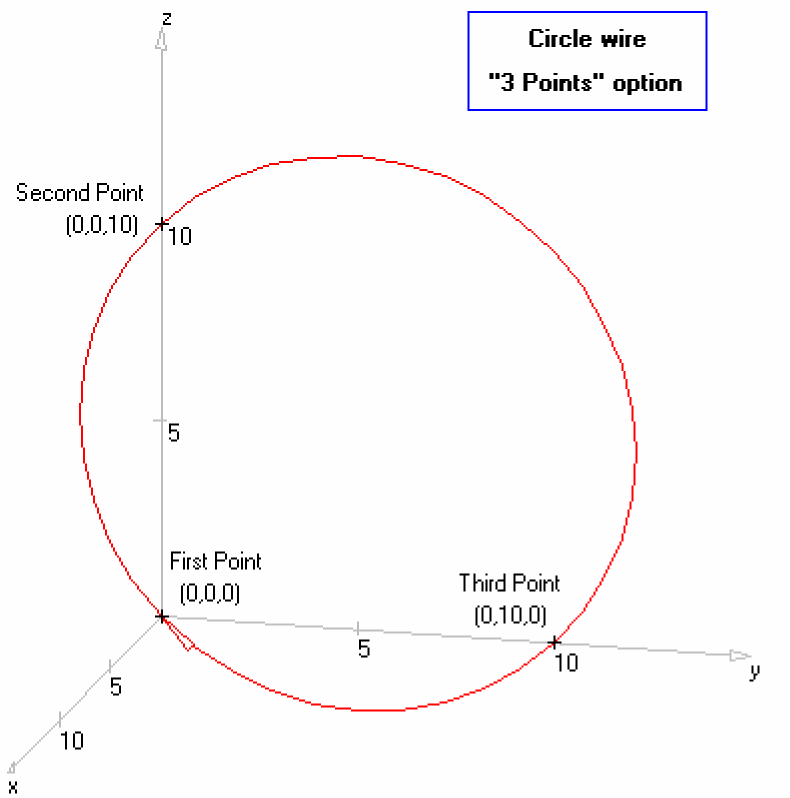


Fig. 5.16: A Circle drawn using the "3 Points" option.

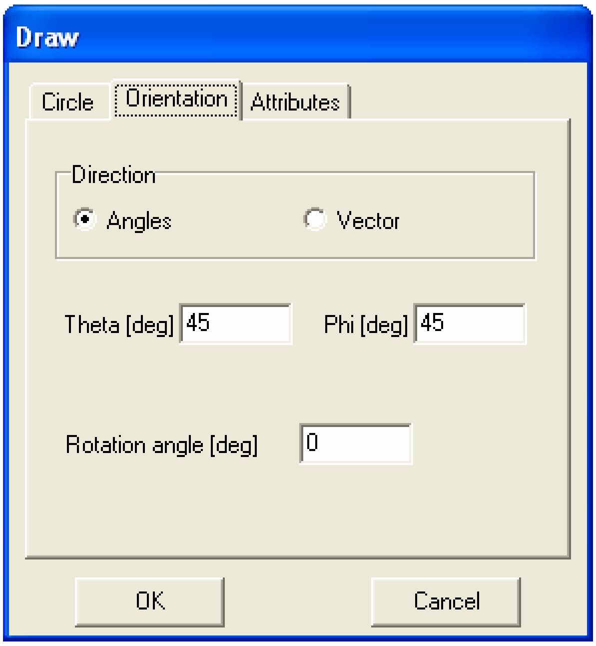


Fig. 5.17: "Angles" option in the Orientation page of the Draw dialog box.

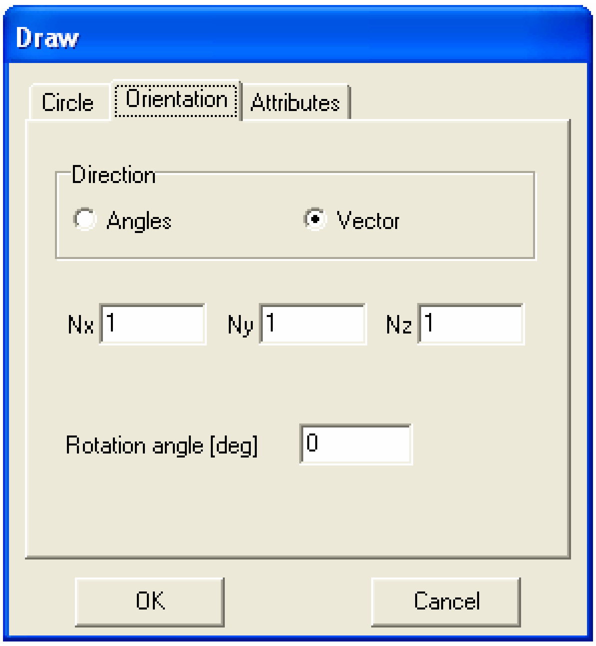


Fig. 5.18: "Vector" option in the Orientation page of the Draw dialog box.

5.4 Helix

The Helix refers to a helical wire.
Choose Draw/Wire/Helix in the main menu to display the Draw dialog box for the Helix, Fig. 5.19. This dialog box has three pages: Helix, Orientation and Attributes, Fig. 5.20.

The Helix page

The Helix page sets the geometrical parameters for the Helix. In this page there are two options: Start - Radius - Pitch - Turns and Start - End - Radius - Turns.

The **Start - Radius - Pitch - Turns** option allows entering the Helix by giving its Start Point, Radius, Pitch and Number of turns, Figs. 5.20 and 5.21. If Pitch is positive the helix will be right-handed, and if Pitch is negative the helix will be left-handed. The helix axis can be defined in the Orientation page.
If **Start - End - Radius - Turns** is chosen, the helix will be drawn starting from Start Point and ending at End Point, with the given Radius and Number of turns, Figs. 5.22 and 5.23. The Number of turns must be an integer number, if it is positive the helix will be right-handed and if it is negative the helix will be left-handed. The orientation of the helix axis is determined by the starting and ending points. The helix can be rotated around its axis by given the Rotation angle. The Orientation page will be invisible when the Start - End - Radius - Turns option is chosen.

Once the geometrical parameters in the Helix page, and eventually in the Orientation page, have been set the Attributes page can be chosen. Section 5.8 describes the parameters that can be defined in the Attributes page. The wire resistivity and coating can be set in the Materials page described in Section 5.9.

The Orientation page

The Orientation page sets the orientation for the Helix.
In this page there is a box with two options: Angles and Vector.

If **Angles** is selected, the helix axis can be defined by given its direction in 3D space. This direction is determined by the Theta and Phi angles in spherical coordinates, Fig. 5.24.
If **Vector** is selected, the helix axis can be defined by given a vector in the axis direction. Its Nx, Ny, and Nz components define this vector, Fig. 5.25.
The helix can be rotated around its axis by given the Rotation angle, Figs. 5.24 and 5.25.

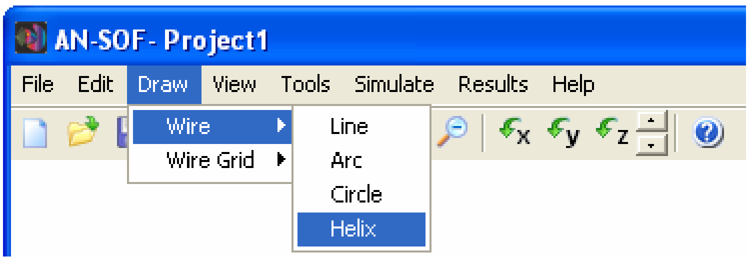


Fig. 5.19: The Draw/Wire/Helix command in the main menu displays the Draw dialog box for the Helix.

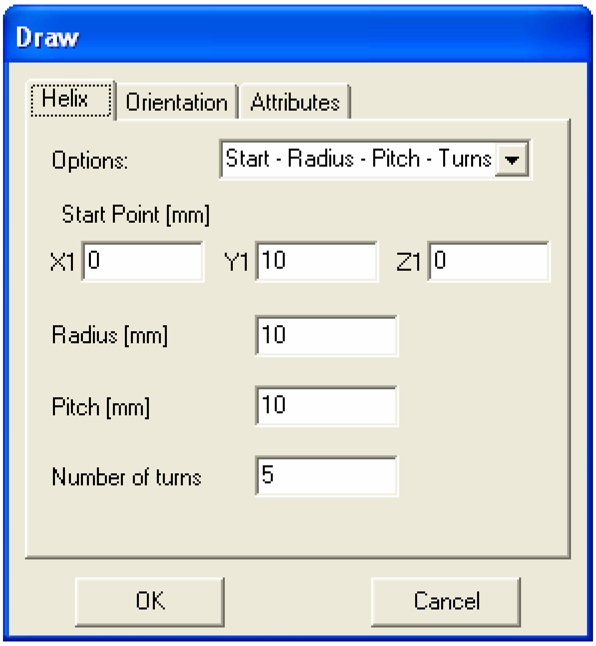


Fig. 5.20: "Start - Radius - Pitch - Turns" option in the Helix page of the Draw dialog box.

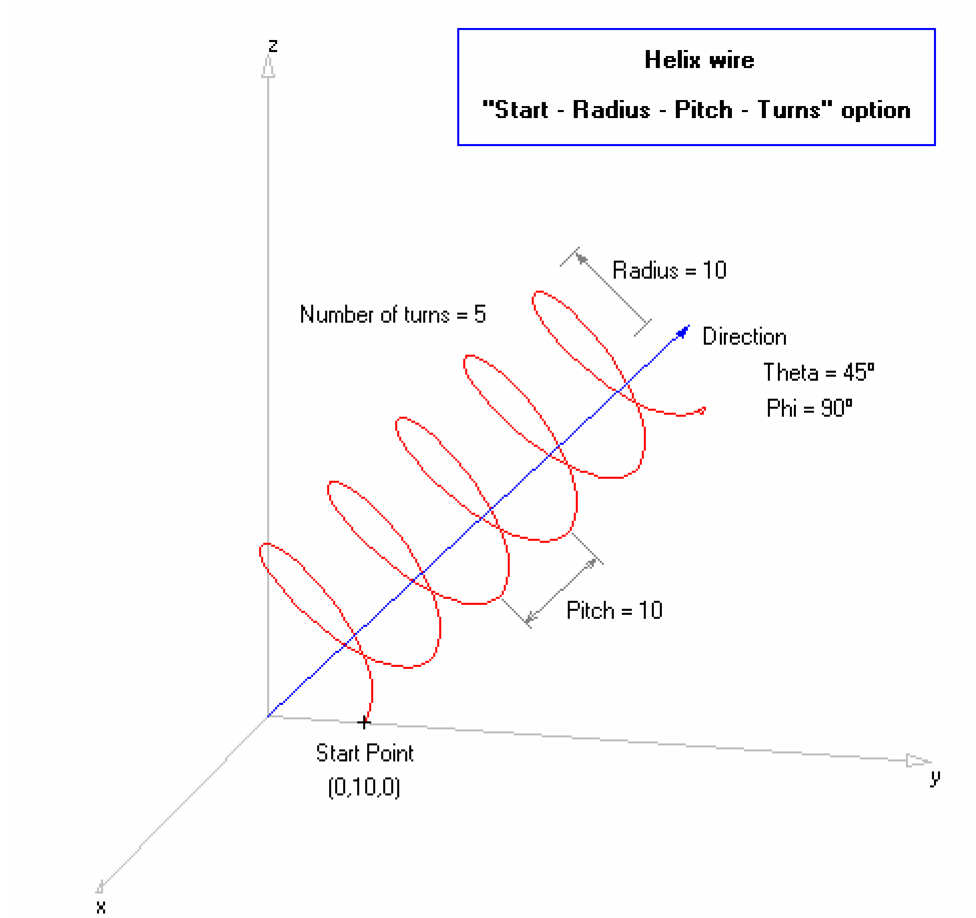


Fig. 5.21: A Helix drawn using the "Start - Radius - Pitch - Turns" option.

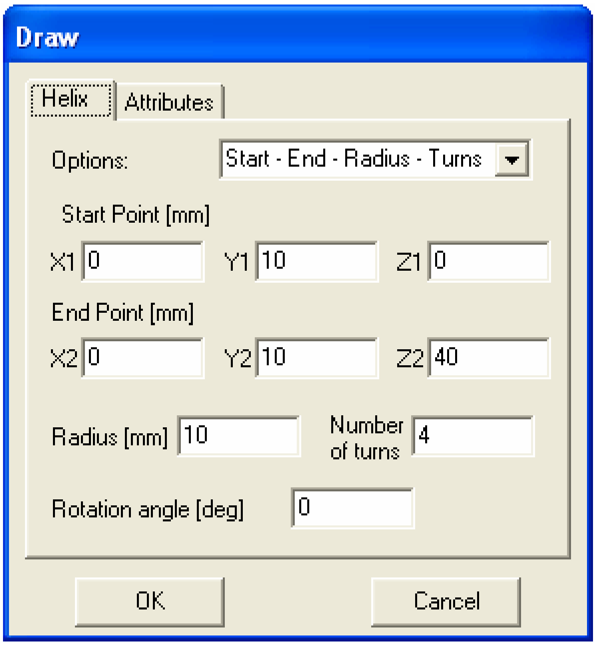


Fig. 5.22: "Start - End - Radius - Turns" option in the Helix page of the Draw dialog box.

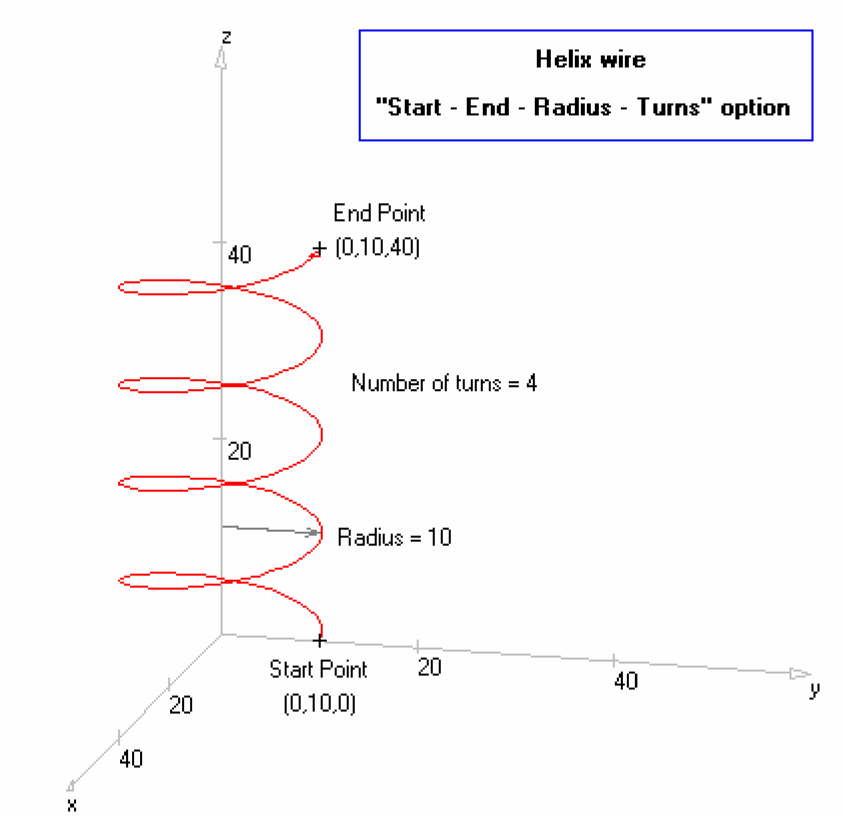


Fig. 5.23: A Helix drawn using the "Start - End - Radius - Turns" option.

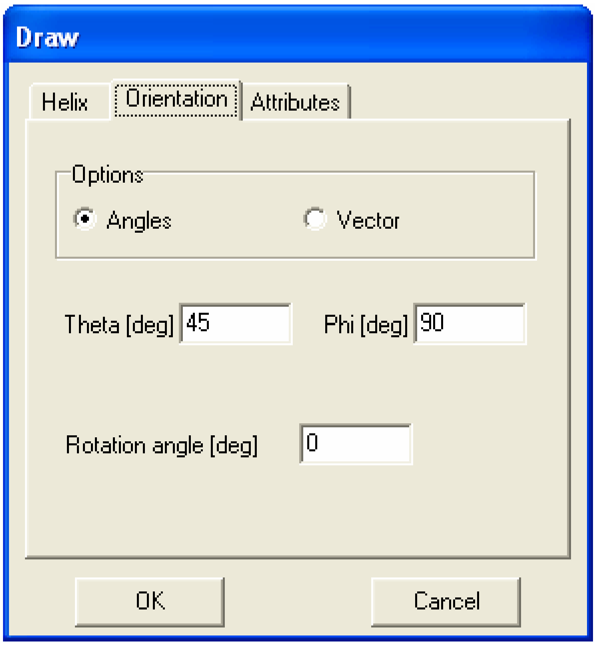


Fig. 5.24: "Angles" option in the Orientation page of the Draw dialog box.

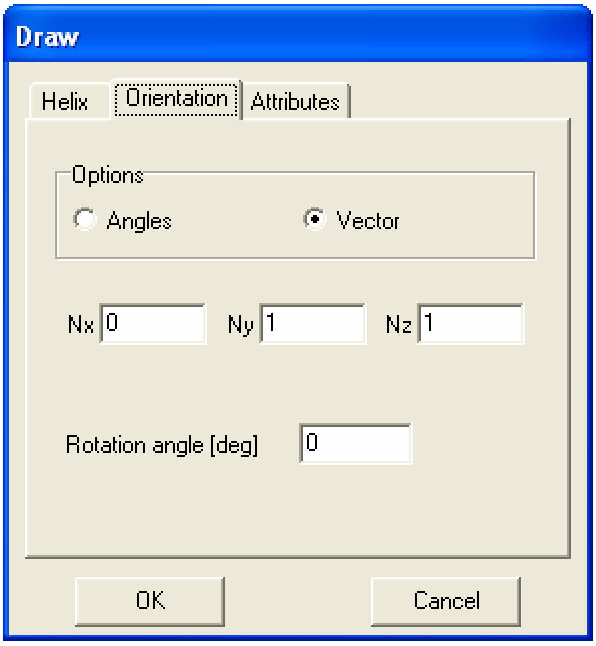


Fig. 5.25: "Vector" option in the Orientation page of the Draw dialog box.

5.5 Quadratic

The Quadratic refers to a quadratic wire or parabola.

Choose Draw/Wire/Quadratic in the main menu to display the Draw dialog box for the Quadratic, Fig. 5.26. This dialog box has two pages: Quadratic and Attributes, Fig. 5.27.

The Quadratic page

The Quadratic page sets the geometrical parameters for the Quadratic.

The Quadratic is entered by giving three points. A quadratic curve starting from Start Point, passing through Second Point and ending at End Point will be drawn on the workspace, as shown in Figs. 5.28.

Once the geometrical parameters in the Quadratic page have been set, the Attributes page can be chosen. Section 5.8 describes the parameters that can be defined in the Attributes page. The wire resistivity and coating can be set in the Materials page described in Section 5.9.

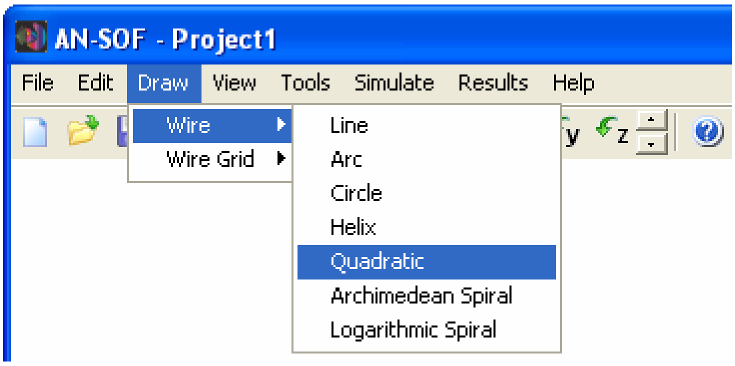


Fig. 5.26: The Draw/Wire/Quadratic command in the main menu displays the Draw dialog box for the Quadratic.

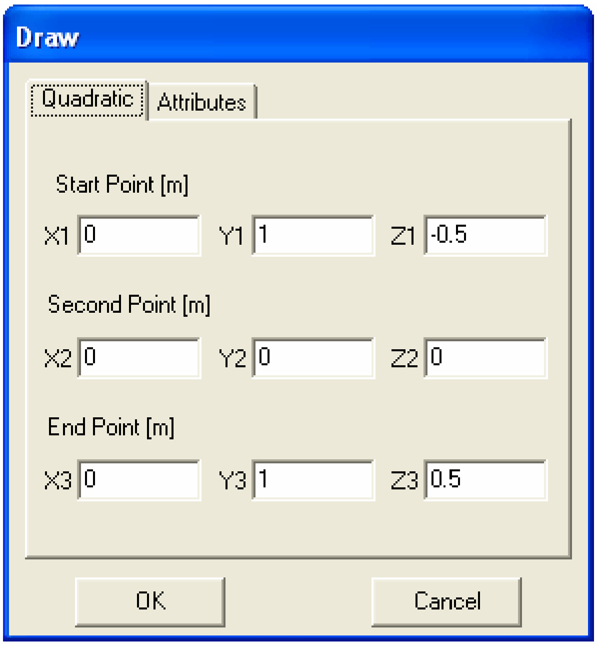


Fig. 5.27: Quadratic page of the Draw dialog box.

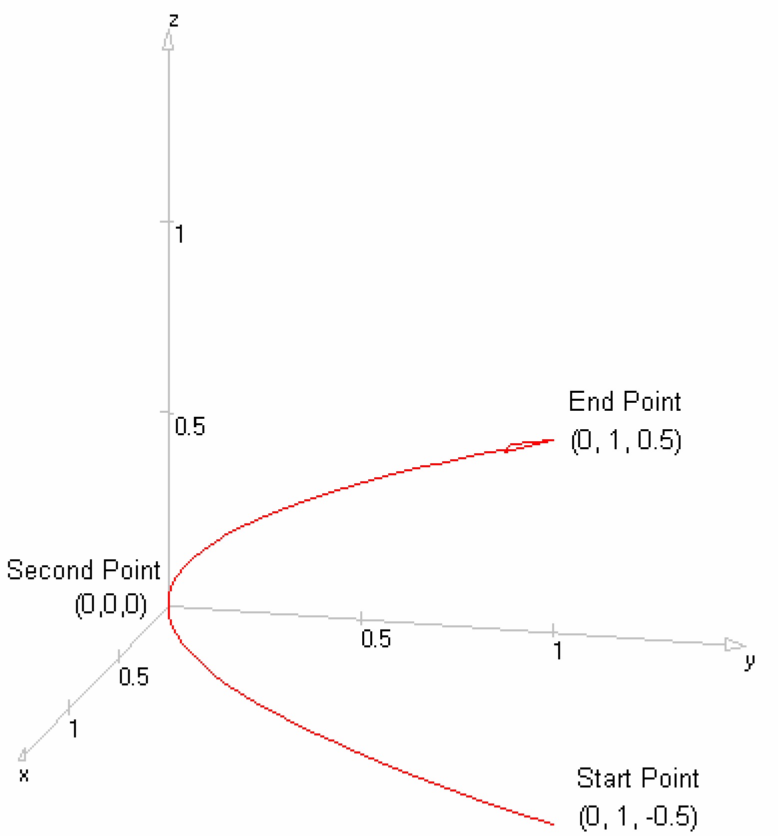


Fig. 5.28: A Quadratic drawn using the points shown in Fig. 5.27.

5.6 Archimedean Spiral

The Archimedean Spiral refers to the Archimedes' spiral with polar equation $r(\alpha) = r_0 + p/(2\pi)\ \alpha$, where r_0 is the starting radius and p is the pitch. For an spiral with an integer number of turns, M, we have $\alpha = 2\pi M$ at its end point, so $r_{end} = r_0 + pM$, the pitch p being the separation between turns. Besides, we have that the pitch equals the constant growth rate of the spiral radius $r(\alpha)$ per turn, that is $p = 2\pi dr/d\alpha$.

Choose Draw/Wire/Archimedean Spiral in the main menu to display the Draw dialog box for the Archimedean Spiral, Fig. 5.29. This dialog box has two pages: Archimedean Spiral and Attributes, Fig. 5.30.

The Archimedean Spiral page

The Archimedean Spiral page sets the geometrical parameters for the Archimedean Spiral.

The Archimedean spiral is entered by giving the Start Point, Start Radius r_0, Pitch p (positive or negative) and Number of Turns M (complete turns and fractions of a turn can be defined). The spiral lies on a plane described by the Orientation Angles Theta and Phi (normal to the plane in spherical coordinates) and can be rotated by defining a Rotation Angle, Fig. 5.31.

Once the geometrical parameters in the Archimedean Spiral page have been set, the Attributes page can be chosen. Section 5.8 describes the parameters that can be defined in the Attributes page. The wire resistivity and coating can be set in the Materials page described in Section 5.9.

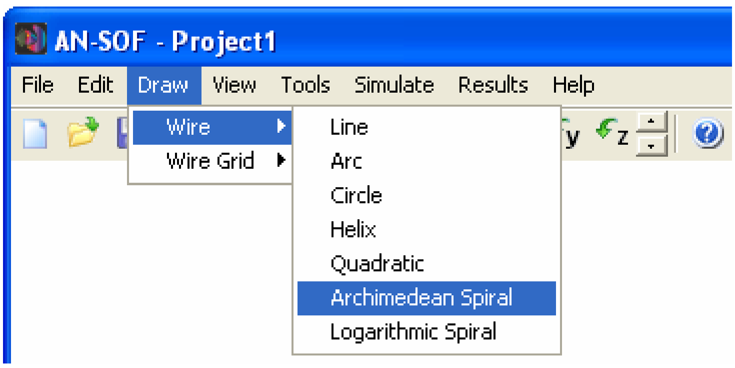


Fig. 5.29: The Draw/Wire/Archimedean Spiral command in the main menu displays the Draw dialog box for the Archimedean Spiral.

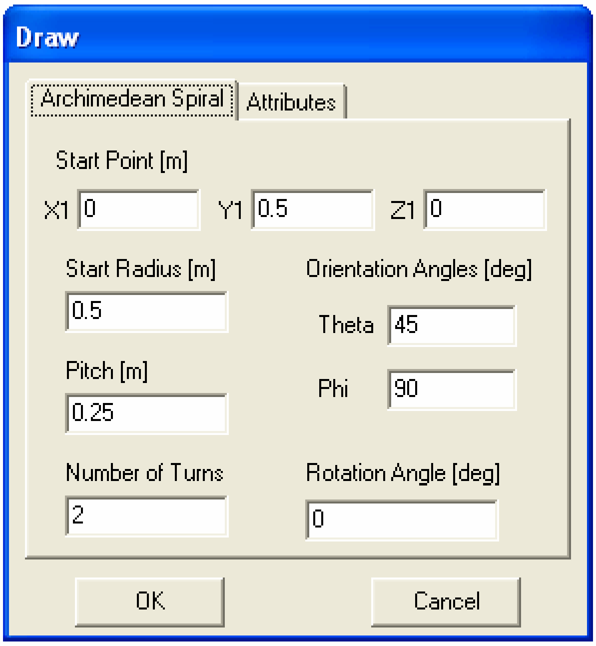


Fig. 5.30: Archimedean Spiral page of the Draw dialog box.

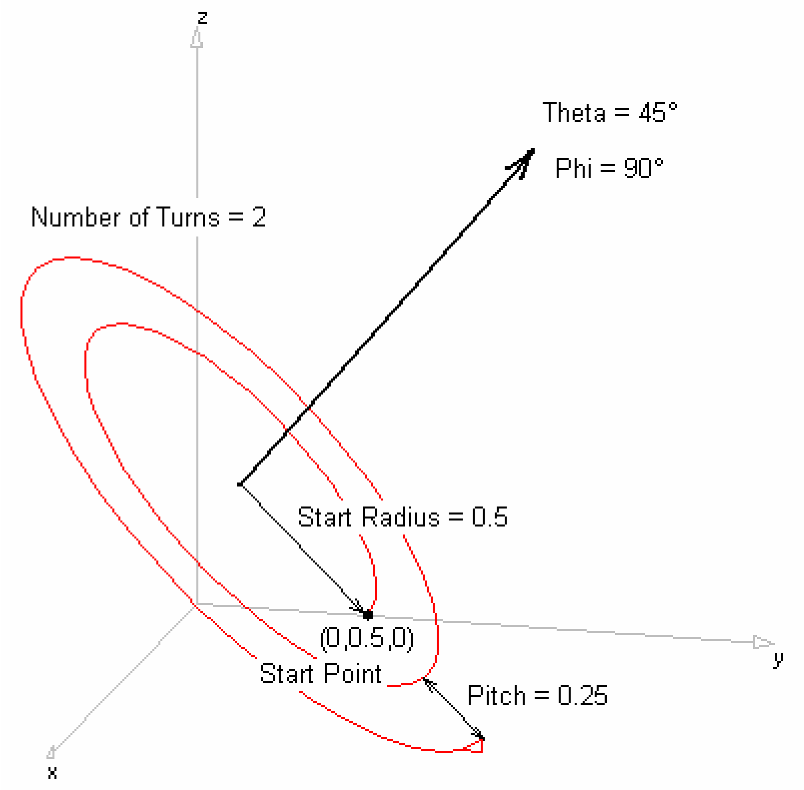


Fig. 5.31: An Archimedean Spiral drawn using the data shown in Fig. 5.30.

5.7 Logarithmic Spiral

The Logarithmic Spiral refers to a spiral with polar equation $r(\alpha) = r_0 \exp(b\alpha)$, where r_0 is the starting radius (r at $\alpha = 0$), $b = p/(2\pi r_0)$ and p is the starting pitch, that is, the derivative $2\pi dr/d\alpha$ at $\alpha = 0$ (starting growth rate of the spiral radius $r(\alpha)$ per turn). The first two terms of the Taylor expansion $r(\alpha) = r_0 + p/(2\pi)\ \alpha + r_0(b\alpha)^2/2 + \ldots$ give the polar equation of an Archimedean spiral, which is described in Section 5.6.

Choose Draw/Wire/Logarithmic Spiral in the main menu to display the Draw dialog box for the Logarithmic Spiral, Fig. 5.32. This dialog box has two pages: Logarithmic Spiral and Attributes, Fig. 5.33.

The Logarithmic Spiral page

The Logarithmic Spiral page sets the geometrical parameters for the Logarithmic Spiral.

The logarithmic spiral is entered by giving the Start Point, Start Radius r_0, Start Pitch p (positive or negative) and Number of Turns (complete turns and fractions of a turn can be defined). The spiral lies on a plane described by the Orientation Angles Theta and Phi (normal to the plane in spherical coordinates) and can be rotated by defining a Rotation Angle, Fig. 5.34.

Once the geometrical parameters in the Logarithmic Spiral page have been set, the Attributes page can be chosen. Section 5.8 describes the parameters that can be defined in the Attributes page. The wire resistivity and coating can be set in the Materials page described in Section 5.9.

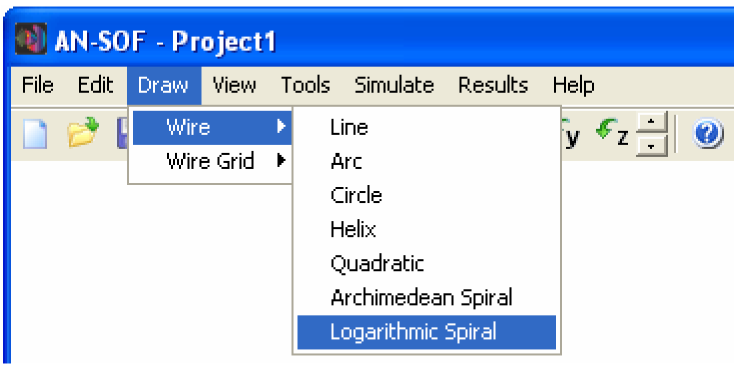


Fig. 5.32: The Draw/Wire/Logarithmic Spiral command in the main menu displays the Draw dialog box for the Logarithmic Spiral.

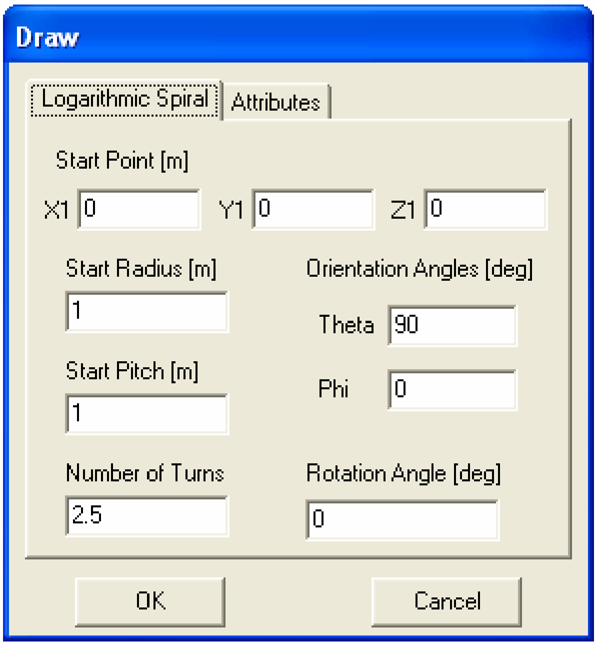


Fig. 5.33: Logarithmic Spiral page of the Draw dialog box.

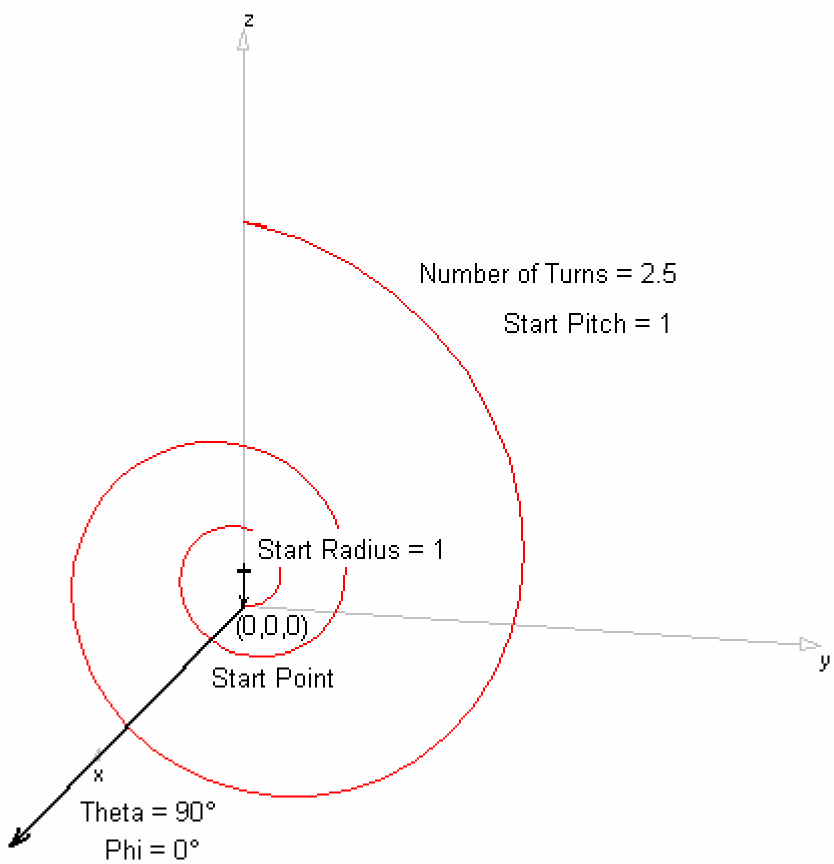


Fig. 5.34: A Logarithmic Spiral drawn using the data shown in Fig. 5.33.

5.8 Wire Attributes

The Attributes page belongs to the Draw dialog box of the chosen wire type, Fig. 5.35. In the Attributes page the following attributes can be specified:

Number of Segments

Any wire has to be divided into a given number of segments. An unknown current on each segment must be found in the simulation process. A default Number of Segments will be shown when the Attributes page is chosen. This number is obtained from the ratio between the wire length and the shortest wavelength, but this value can be modified by the user.
If the **Number of Segments is set to zero**, the program will compute the number of segments consistent with the highest frequency or shortest wavelength.

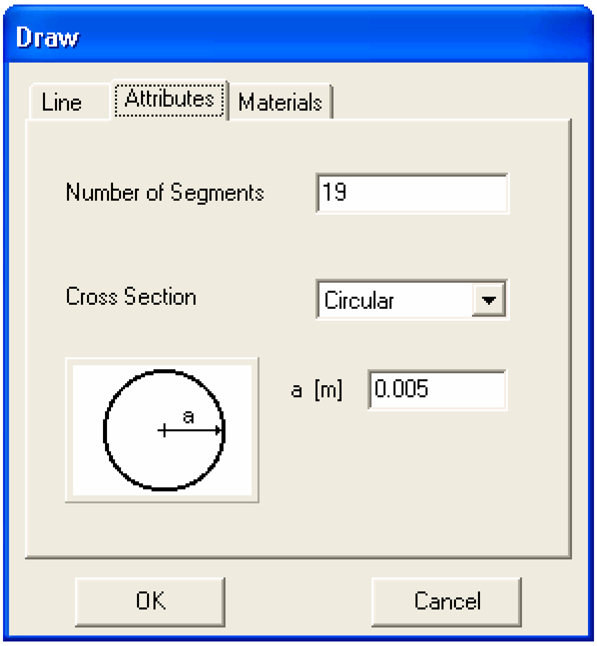


Fig. 5.35: Attributes page in the Draw dialog box for the Line wire.

Cross-Section

The Cross-Section of the wire can be chosen from a combo-box.

There are five cross-section types available: Circular, Square, Flat, Elliptical and Rectangular. The program computes an equivalent radius for the four last cases.

Infinitesimally thin wires are not allowed, so the cross-section radius must be greater than zero.

The Draw dialog box for any wire type has its own Attributes page with the same features described here.

5.9 Wire Materials

The Materials page belongs to the Draw dialog box of the chosen wire type, Fig. 5.36. In the Materials page the following attributes can be specified:

Wire Resistivity [Ohm m]

A resistivity in [Ohms meter] can be specified for the wire. This value is used for computing a distributed impedance along the wire, taking into account the *skin effect*. The equivalent radius for wires of non-circular cross section will be used to compute the impedance per unit length along the wires.
Resistivity values in [Ohms meter] for most common conductive materials are the following:

Copper
1.74E-8

Aluminum (6061-T6)
4E-8

Tin
1.14E-7

Zinc
6E-8

The resistivity of wires is taken into account in the computation if the option Wire Resistivity is checked in the Options page of the Configuration dialog box.

Wire Coating

Wires can have an insulation or coating material. The cross section of a coated wire is considered to be circular, so the equivalent radius will be used for wires having a non-circular cross section. In this case, the material the coating is made of can be defined by the following parameters:

Relative Permittivity
It is the permittivity or dielectric constant of the coating material relative to the permittivity of vacuum.

Relative Permeability
It is the magnetic permeability of the coating material relative to the permeability of vacuum.

Thickness
It is the thickness of the coating shield. It can be set to zero when no coating is used.

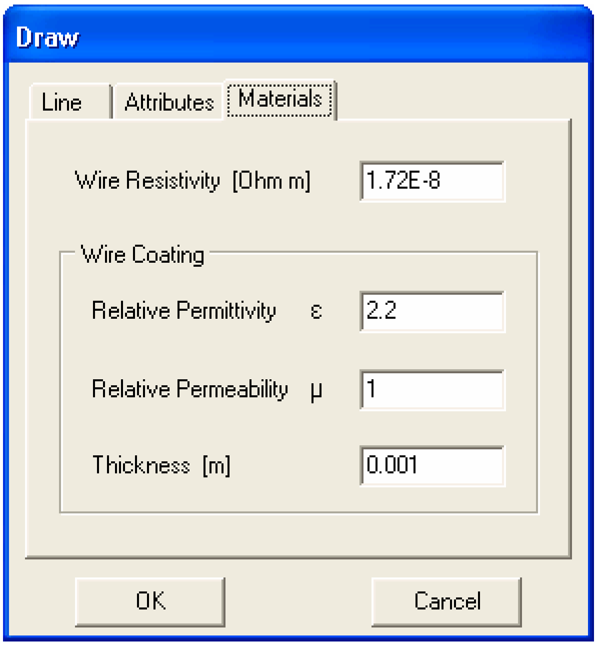


Fig. 5.36: Materials page in the Draw dialog box for the Line wire.

5.10 Enabling/Disabling Resistivities

If wires with non-zero resistivity have been drawn previously and the whole structure must now be considered as a perfect electric conductor, all resistivities can be disabled without modifying the definitions of the wires.

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Options page, Fig. 5.37. If the option Wire Resistivity in this page is checked, the resistivities are enabled. Uncheck the Wire Resistivity option in order to disable all of them.

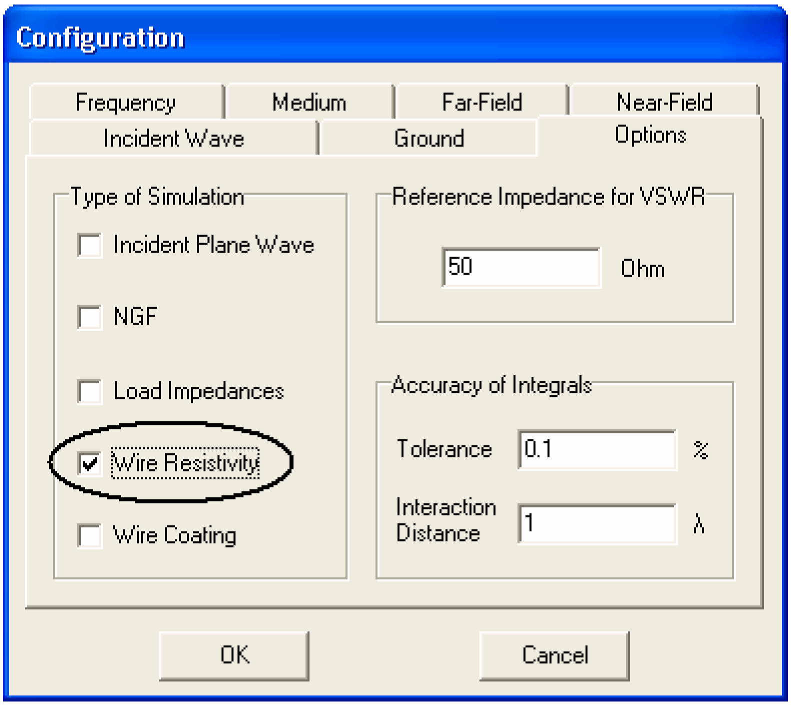


Fig. 5.37: Wire Resistivity option in the Options page of the Configuration dialog box. If this option is checked, all resistivities are enabled, otherwise they are disabled.

5.11 Enabling/Disabling Coatings

If wires with a coating shield or insulation have been drawn previously and the whole structure must now be considered as composed of bare conductive wires, all coatings can be disabled without modifying the definitions of the wires.

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Options page, Fig. 5.38. If the option Wire Coating in this page is checked, the coatings are enabled. Uncheck the Wire Coating option in order to disable all of them.

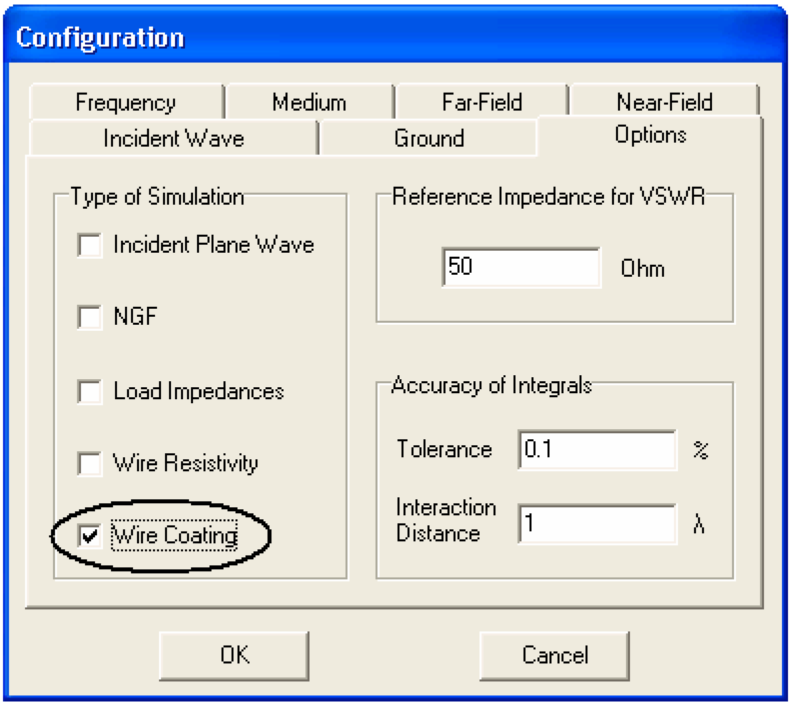


Fig. 5.38: Wire Coating option in the Options page of the Configuration dialog box. If this option is checked, all coatings are enabled, otherwise they are disabled.

5.12 Cross-Section Equivalent Radius

The wire cross-section can be chosen from a combo-box in the Attributes page of the wire Draw dialog box, Fig. 5.39.

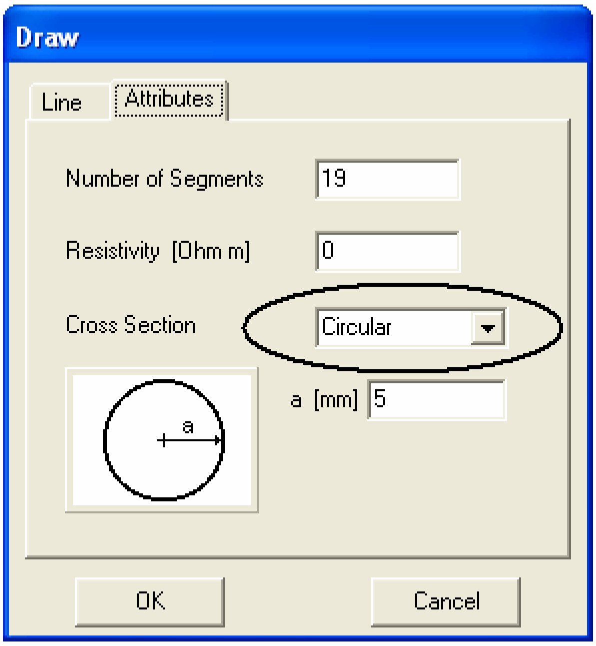
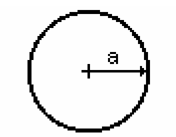


Fig. 5.39: Cross-section combo-box in the Attributes page of the Draw dialog box. A circular cross section of radius "a" is chosen.

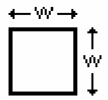
There are five cross-section types available: Circular, Square, Flat, Elliptical and Rectangular. AN-SOF® computes an equivalent radius for the non-circular cross-sections. This equivalent radius is the radius of a circular cross-section that produces the same average electromagnetic fields around the wire and on its surface.

The cross-sections and their equivalent radii are:



Circular

A positive and non-zero radius "a" must be defined. The equivalent radius is the same "a".

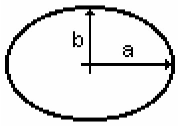


Square

A positive and non-zero width “w” must be defined. The equivalent radius is equal to 0.59017 w.

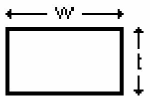
Flat

A positive and non-zero width “w” must be defined. The equivalent radius is equal to w/4.



Elliptical

The semi-axes “a” and “b” must be positive and non-zero. The equivalent radius is equal to (a + b)/2.



Rectangular

The widths “w” and “t” must be positive and non-zero. The equivalent radius is computed by the program using a polynomial and logarithmic approximation to the solution of an integral equation.

5.13 Importing Wires

A list of linear wires written in a file in text (ASCII) format can be imported to AN-SOF by choosing File/Import Wires in the main menu, Fig. 5.40. A sub-menu having three options will be displayed: NEC, MM, and EZ formats.

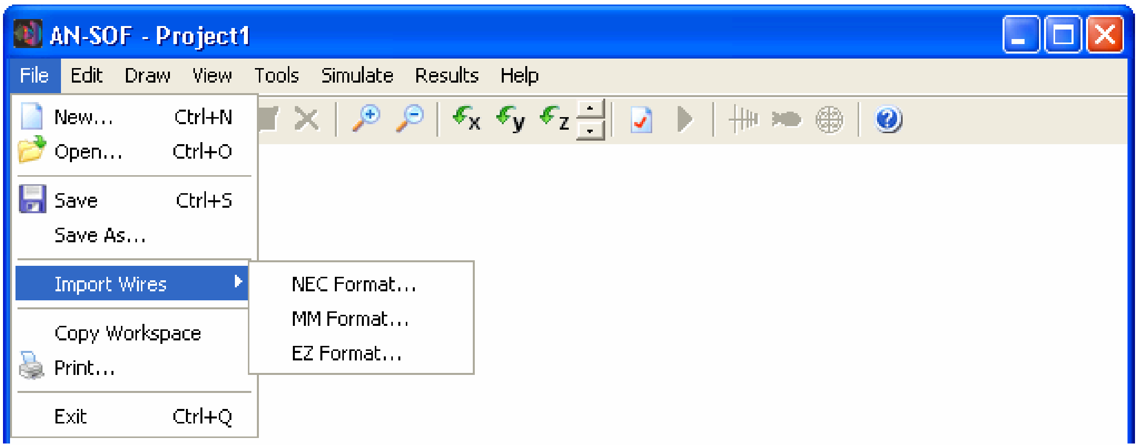


Fig. 5.40: File/Import Wires option in the main menu.

NEC format
One wire per line have to be defined beginning with “GW” as follows:

```
GW LineTag Segments X1 Y1 Z1 X2 Y2 Z2 Radius
[Enter]
```

where

`LineTag` = Tag for the line in the text file. It will be ignored. The space between “GW” and LineTag is optional.
`Segments` = Number of segments for the wire.
`X1 Y1 Z1` = Cartesian coordinates of the starting point for the linear wire.
`X2 Y2 Z2` = Cartesian coordinates of the ending point for the linear wire.
`Radius` = Radius of the wire.

Spaces between fields can be replaced by commas and more than one space can be used. The text lines above the GW lines will be ignored, so comments can be added at the beginning of the file. The last GW line must end with an Enter (press Enter in the keyboard for a carriage return).

The following are equivalent examples:

```
Write comments here
GW 1 12 5.42 0.38 1.262 5.425 -0.378 1.261 0.01
GW 2 5 7.45 0 1.122 7.45 0 1.49 0.015
GW 3 2 8.3 0.0 1.12 8.37 0.0 1.595 0.01
[Enter]
```

```
Write comments here
GW1,12,5.42,0.38,1.262,5.425,-0.378,1.261,0.01
GW2,5,7.45,0,1.122,7.45,0,1.49,0.015
GW3,2,8.3,0.0,1.12,8.37,0.0,1.595,0.01
[Enter]
```

```
Write comments here
GW 1, 12, 5.42, 0.38, 1.262, 5.425, -0.378, 1.261, 0.01
GW 2, 5, 7.45, 0, 1.122, 7.45, 0, 1.49, 0.015
GW 3, 2, 8.3, 0.0, 1.12, 8.37, 0.0, 1.595, 0.01
[Enter]
```

MM format
One wire per line have to be defined as follows:

```
X1,[TAB]Y1,[TAB]Z1,[TAB]X2,[TAB]Y2,[TAB]Z2,[TAB]Radius,
[TAB]Segments
[Enter]
```

where

`X1 Y1 Z1` = Cartesian coordinates of the starting point for the linear wire.
`X2 Y2 Z2` = Cartesian coordinates of the ending point for the linear wire.
`Radius` = Radius of the wire.
`Segments` = Number of segments for the wire.

The last text line must end with an Enter (press Enter in the keyboard for a carriage return).

Example:

```
5.42, 0.38, 1.262,  5.425, -0.378,  1.261, 0.01,   12
7.45, 0,    1.122,  7.45,  0,       1.49,  0.015,  5
8.3,  0.0,  1.12,   8.37,  0.0,     1.595, 0.01,   2
[Enter]
```

EZ format
One wire per line have to be defined as follows:

```
LineTag(13)X1(7),Y1(7),Z1(7)Tag(10)X2(7),Y2(7),Z2(7)Spa
ce(1)Diameter(9)Space(3)Segments(4)Space(3)Permittivity
(6)Space(1)Thickness(8)[Enter]
```

where

`LineTag(13)` = Tag for the line in the text file. It must be 13 characters long. It will be ignored.

`X1(7),Y1(7),Z1(7)` = Cartesian coordinates of the starting point for the linear wire. Each one must be 7 characters long. Commas are used to separate the coordinates.
`Tag(10)` = Separation tag. It must be 10 characters long. Spaces can be used.
`X2(7),Y2(7),Z2(7)` = Cartesian coordinates of the ending point for the linear wire. Each one must be 7 characters long. Commas are used to separate the coordinates.
`Space(1)` = Space. One character long.
`Diameter(9)` = Diameter of the wire. It must be 9 characters long. American wire gauge (AWG) can be used in the format “#n”, where n is the gauge. Negative values of n indicates AWG = 00, 000, 0000, etc., that is, #-1, #-2, #-3. etc.
`Space(3)` = Space. Three characters long.
`Segments(4)` = Number of segments for the wire. It must be 4 characters long.
`Space(3)` = Space. Three characters long.
`Permittivity(6)` = Relative permittivity or dielectric constant of the wire coating material. It must be 6 characters long.
`Space(1)` = Space. One character long.
`Thickness(8)` = Thickness of the wire coating material. It must be 8 characters long.

The last text line must end with an Enter (press Enter in the keyboard for a carriage return). Due to the constant character length of each field, spaces can be used to complete the string.

Example:

```
1      START  -2.6667,      2,4.66667    END         -2,
5,4.66667   .249606   2           1        0
2      START  -3.6667,      0,5.33587    END         -4,
2,4.66667   .249606   2           1        0
3      START  -5.6667,     -2,4.80001    END         -2,
7,3.90538   .249606   2           1        0
[Enter]
```

In the NEC, MM and EZ formats, automatic segmentation of a wire can be obtained by entering any number equal or less than zero as the number of segments.

The units for the coordinates of the starting and ending points of any wire must be consistent with the length unit chosen in the Preferences dialog box. Also, the wire radius or diameter of any imported wire will be considered to be expressed in the unit chosen in the Preferences dialog box.

6. Editing Wires

6.1 Selecting a Wire

Any wire on the workspace can be selected in three different ways:

1. By clicking the left mouse button on it.
2. By clicking the right mouse button on it. In this case, a pop-up menu will be shown, Fig. 6.1.
3. By pressing the left and right arrows on the keyboard.

A wire is highlighted in red when it is selected.

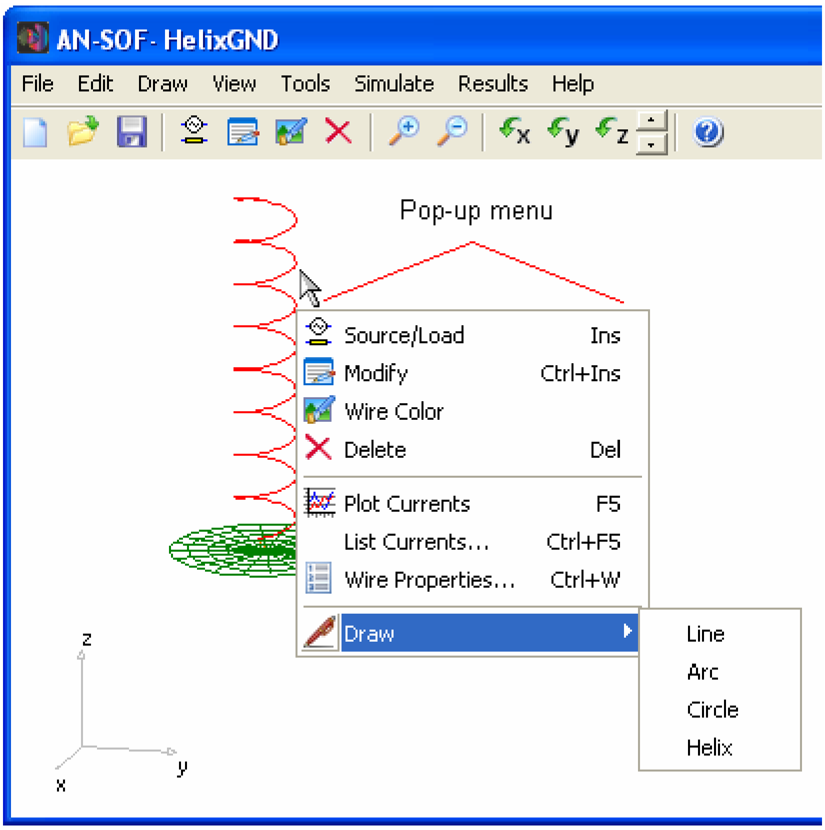


Fig. 6.1: Pop-up menu displayed when a wire is selected by clicking the right mouse button on it.

6.2 The Pop-Up Menu

When a wire is selected by clicking the right mouse button on it, the displayed pop-up menu has the following commands:

Source/Load
Displays the Source/Load toolbar for exciting or loading the selected wire.

Modify
Displays the Modify dialog box for modifying the selected wire.

Wire Color
Displays a Windows® dialog box for changing the color of the selected wire.

Delete
Deletes the selected wire with all sources and loads placed on it.

Plot Currents
Executes the AN-XY Chart® program for plotting the currents vs. position along the selected wire. This command is enabled when the currents are computed.

List Currents
Displays the List Currents toolbar for listing the currents vs. frequency at any segments on the selected wire. This command is enabled when the currents are computed.

Wire Properties
Displays the Wire Properties dialog box for viewing information about the selected wire.

Draw
Contains a sub-menu with the Line, Arc, Circle, Helix, Quadratic, Archimedean Spiral and Logarithmic Spiral commands for drawing these wire kinds.

6.3 Modifying a Wire

Clicking the right mouse button on a wire shows a pop-up menu, Fig. 6.1. Choosing the Modify command from the pop-up menu shows the Modify dialog box, where the geometrical parameters and attributes of the selected wire can be modified.

The Modify command can also be chosen by first selecting a wire by clicking the left mouse button on it, and next choosing Edit/Modify in the main menu, Fig. 6.2.

When a wire is modified, all sources and loads placed on it are removed.

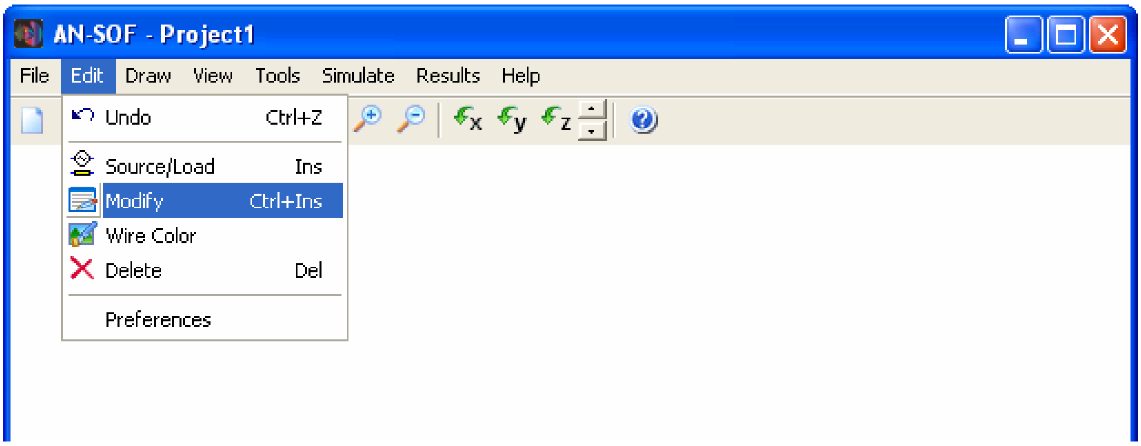


Fig. 6.2: Modify command in the Edit menu. This command is enabled when a wire is selected.

6.4 Deleting a Wire

Clicking the right mouse button on a wire shows a pop-up menu, Fig. 6.1. Choosing the Delete command from the pop-up menu deletes the selected wire with all sources and loads placed on it.

The Delete command can also be chosen by first selecting a wire by clicking the left mouse button on it, and next choosing Edit/Delete in the main menu, Fig. 6.3.

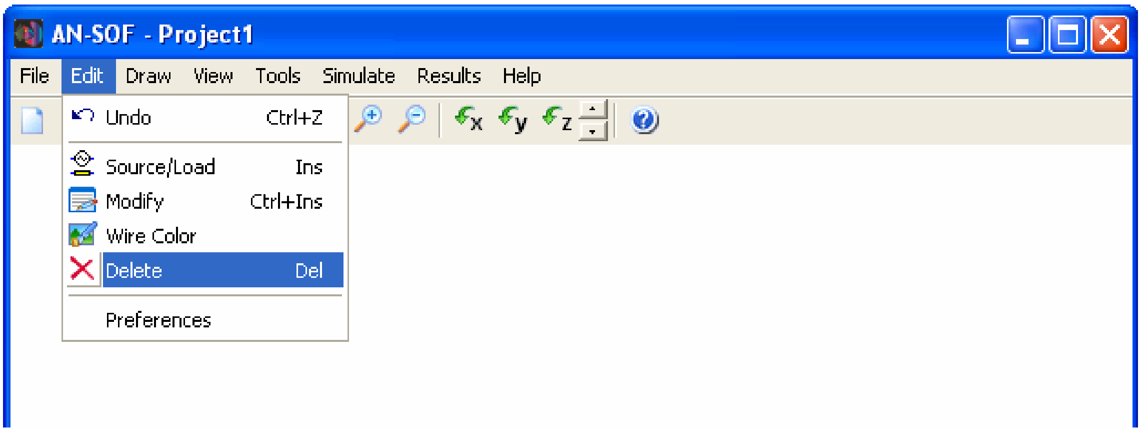


Fig. 6.3: Delete command in the Edit menu. This command is enabled when a wire is selected.

6.5 Deleting a Group of Wires

Clicking the left mouse button on the workspace, while maintaining pressed the Ctrl key, a selecting box can be expanded, Fig. 6.4. This selecting box permits selecting a group of wires in the project workspace, which is highlighted in red.

Choosing the Edit/Delete command in the main menu deletes the selected group of wires.

The Delete command can also be executed by pressing Del on the keyboard or the Edit toolbar.

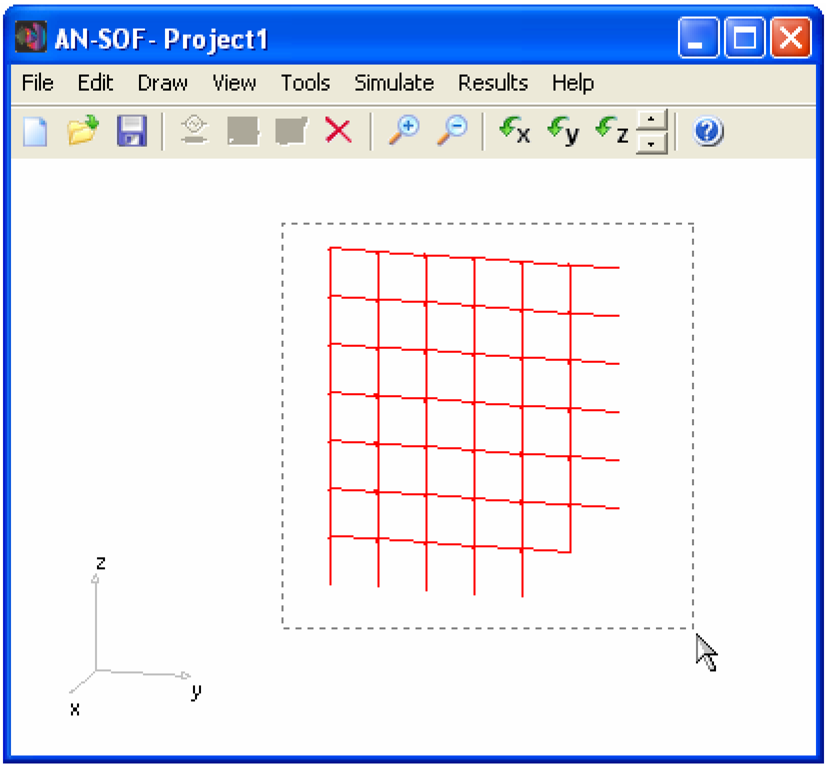


Fig. 6.4: Box for selecting a group of wires. The left mouse button and Ctrl key must be pressed at the same time.

6.6 Wire Color

Clicking the right mouse button on a wire shows a pop-up menu, Fig. 6.1. Choosing the Wire Color command from the pop-up menu displays a Windows® dialog box for changing the color of the selected wire. This command is enabled when a wire is selected.

The Wire Color command can also be chosen by first selecting a wire by clicking the left mouse button on it, and next choosing Edit/Wire Color in the main menu, Fig. 6.5.

The color of a group of wires can be changed by first selecting the wires and next pressing Edit/Wire Color in the main menu. A group of wires can be selected by expanding a selecting box as explained in the previous section.

Fig. 6.5: Wire Color command in the Edit menu. This command is enabled when a wire is selected.

6.7 Viewing Wire Properties

Clicking the right mouse button on a wire will display a pop-up menu, Fig. 6.6, where the Wire Properties... command can be selected.

The Wire Properties... command can also be chosen by first selecting a wire by clicking the left mouse button on it, and next choosing View/Wire Properties... in the main menu, Fig. 6.7.

Choose the Wire Properties... command to display the Wire Properties dialog box, Fig. 6.8.

The Wire Properties dialog box has two pages: Geometrical and Electrical.

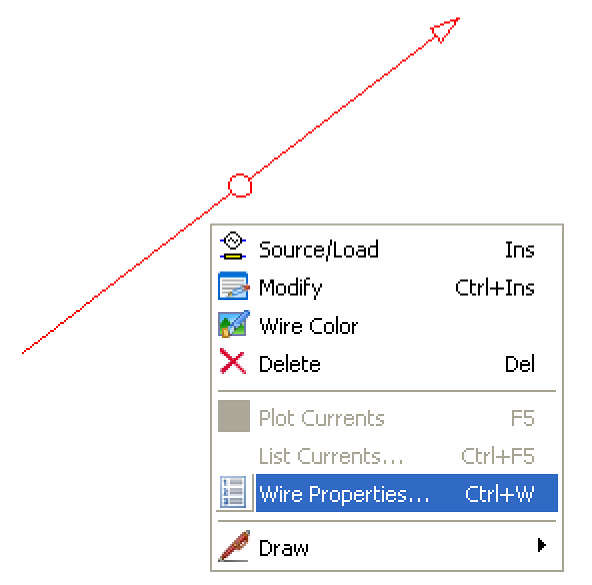


Fig. 6.6: Wire Properties... command in the pop-up menu.

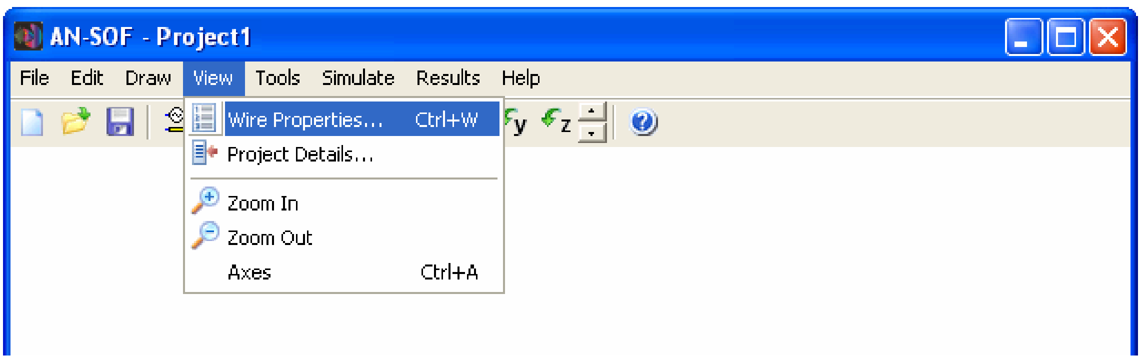


Fig. 6.7: Wire Properties... command in the main menu.

The Geometry page

Shows the geometrical properties of the selected wire, Fig. 6.8.
These properties are:

- **Start Point:** Shows the starting point coordinates of the selected wire.
- **End Point:** Shows the ending point coordinates of the selected wire.
- **Length:** Shows the wire length.
- **Longest Segment:** Shows the length of the longest segment.
- **Shortest Segment:** Shows the length of the shortest segment.
- **Shortest Wavelength λ:** Shows the wavelength related to the highest frequency.
- **Length/λ:** Shows the wire length in wavelengths. The wavelength corresponds to the highest frequency.
- **Longest Segment/λ:** Shows the length of the longest wire segment in wavelengths. The wavelength corresponds to the highest frequency.
- **Shortest Segment/λ:** Shows the length of the shortest wire segment in wavelengths. The wavelength corresponds to the highest frequency.

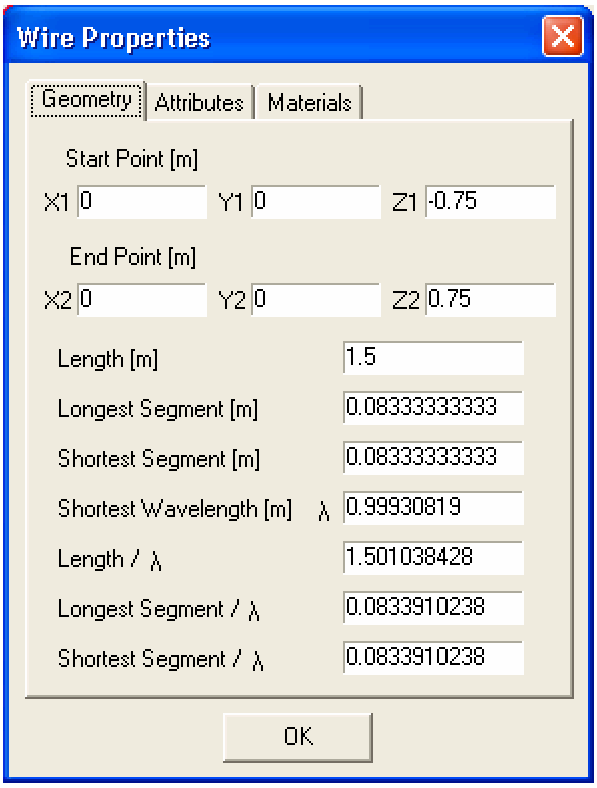


Fig. 6.8: Wire Properties dialog box. The Geometry page shows the geometrical properties of the selected wire.

The Attributes page

Shows the electrical properties of the selected wire, Fig. 6.9.
These properties are:

- **Number of Segments:** Shows the number of segments into which the selected wire has been divided.
- **Number of Sources:** Shows the number of sources placed on the selected wire.
- **Number of Loads:** Shows the number of loads placed on the wire.
- **Cross-Section:** Shows the cross-section type and its dimensions.
- **Thin-Wire ratio:** It is the wire diameter to the shortest segment length ratio and must be less than 1. It is a measure of the wire thinness. If the Thin-Wire ratio is greater than 1, then the thin-wire approximation is not satisfied (see Background Theory), and results after a simulation could be inaccurate. For a non-circular cross-section, the wire diameter is two times the equivalent radius of the cross-section.

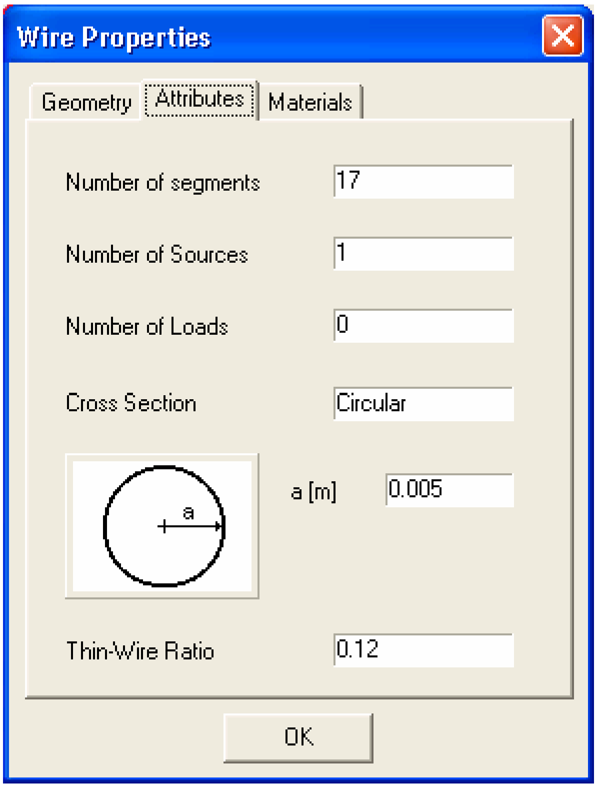


Fig. 6.9: Wire Properties dialog box. The Attributes page shows the segmentation used for the selected wire, the number of sources and loads placed on the wire, and the type of cross section.

The Materials page

Shows the properties of the materials the selected wire is made of, Fig. 6.9. These properties are:

- **Wire Resistivity:** Shows the resistivity of the selected wire in [Ohm m]. If the wire is coated, it is the resistivity of the internal conductor.
- **Wire Coating:** Shows the parameters of the coating shield of the selected wire.
    - **Relative Permittivity:** It is the permittivity or dielectric constant of the coating material relative to the permittivity of vacuum.
    - **Relative Permeability:** It is the magnetic permeability of the coating material relative to the permeability of vacuum.
    - **Thickness:** It is the thickness of the coating shield.

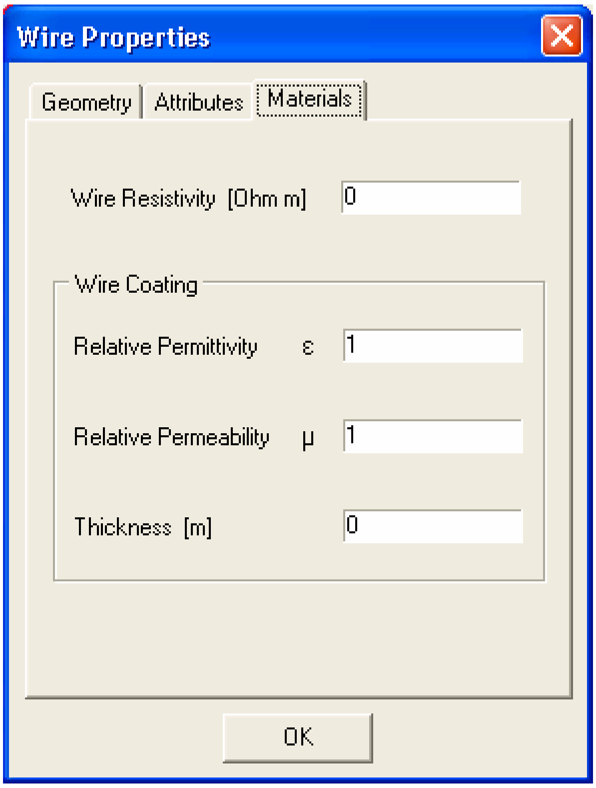


Fig. 6.10: Wire Properties dialog box. The Materials page shows the material parameters of the conductive wire and its coating shield or insulation.

6.8 Connecting Wires

Any wire has two ends: a starting point, called "Start Point", and an ending point, called "End Point".

A wire junction is automatically established whenever the coordinates of a wire end are identical to the end coordinates of a wire previously specified. Wire junctions must be established in order to satisfy Kirchhoff's current law at the connection point.

Wires can also be connected by copying and pasting their ends.

The following example will show how to connect the Start Point of a given wire #1 to the Start Point of a new wire #2.

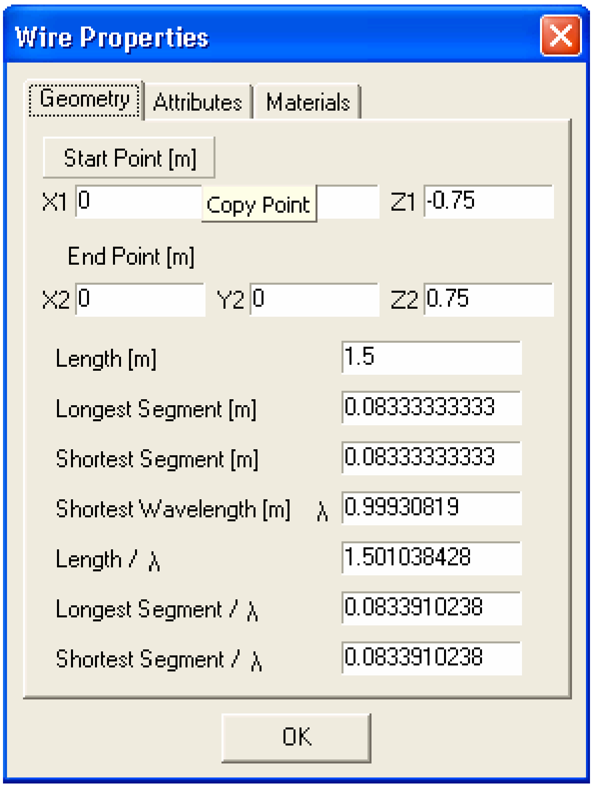


Fig. 6.11: Wire Properties dialog box. The Geometry page shows the geometrical properties of wire #1. Pressing the "Start Point" button will copy the starting point of the wire.

Step-by-step procedure for connecting the wires:

1. Clicking the right mouse button on wire #1 will display a pop-up menu, Fig. 6.6.
2. Choose the Wire Properties... command from the pop-up menu to display the Wire Properties dialog box, Fig. 6.8.
3. The Geometry page of this dialog box shows the geometrical properties of wire #1. Press the “Start Point” button for copying the starting point coordinates of wire #1, Fig 6.11. The ending point coordinates of wire #1 could also be copied by pressing the “End Point” button, Fig. 6.12.
4. In this example, wire #2 will be a Line. Then, choose Draw/Wire/Line in the main menu to display the Draw dialog box for the Line.
5. Press the “From Point” button to paste the copied point, Fig. 6.13. Then, follow the procedure described in Section 5.1 to complete the definition of wire #2.

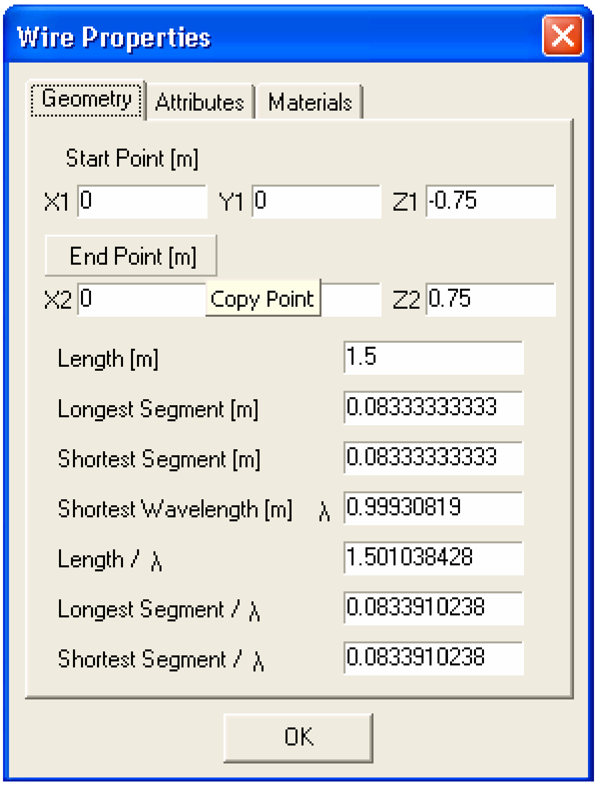


Fig. 6.12: Wire Properties dialog box. The Geometrical page shows the geometrical properties of wire #1. Pressing the “End Point” button will copy the ending point of the wire.

Thus, the starting points of wire #1 and wire #2 will have exactly the same coordinates, so they will be electrically connected.

The user can connect an arbitrary number of wires, of any kind, at the same point by means of this procedure.

If this procedure is not used, two wires are considered to be connected when their ends are “very close” to each other. In this case, “very close” means that wire ends are separated by a distance less than a tenth of the wire radius.

All wires are assumed to have no end caps. End caps can be simulated by adding the radius of the wire to its length on the specified end.

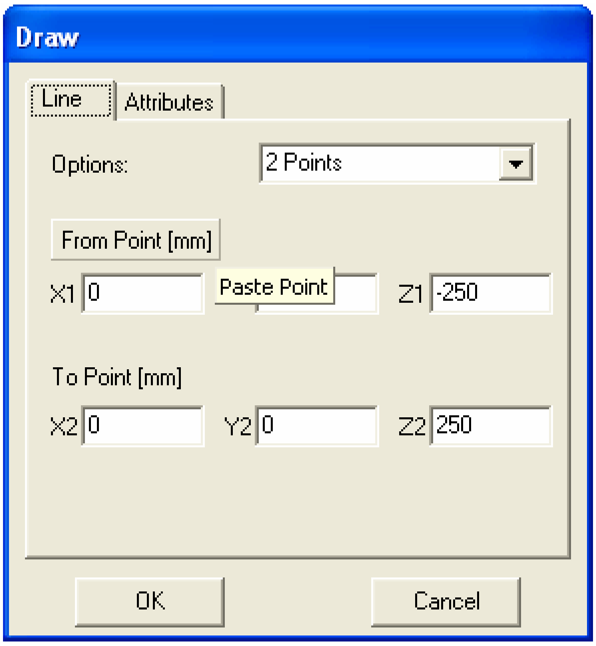


Fig. 6.13: "2 Points" option in the Line page of the Draw dialog box for the Line wire #2. Pressing the “From Point” button will paste the copied point of wire #1.

6.9 Project Details

Choose View/Project Details... in the main menu to display the Project Details dialog box, Figs. 6.14 and 6.15. Details for the project can be viewed by selecting this command.

This information includes:

- **Last saved:** Date and time of the last saved project file (*.EMM file).
- **No. of wires:** Number of wires drawn on the workspace.
- **No. of sources:** Total number of discrete sources placed on the wire structure.
- **No. of loads:** Total number of lumped loads placed on the wire structure.
- **No. of segments:** Total number of segments into which the wire structure has been divided.
- **No. of connections:** Total number of connections on the wire structure.
- **No. of ground points:** Total number of ground connections.
- **Total No. of Unknowns:** Total number of unknown currents to be computed. It is the sum of the number of segments, connections and ground points.

The Project Details dialog box has a window where notes and comments can be written. This text will be saved in a .txt file with the same name of the project.

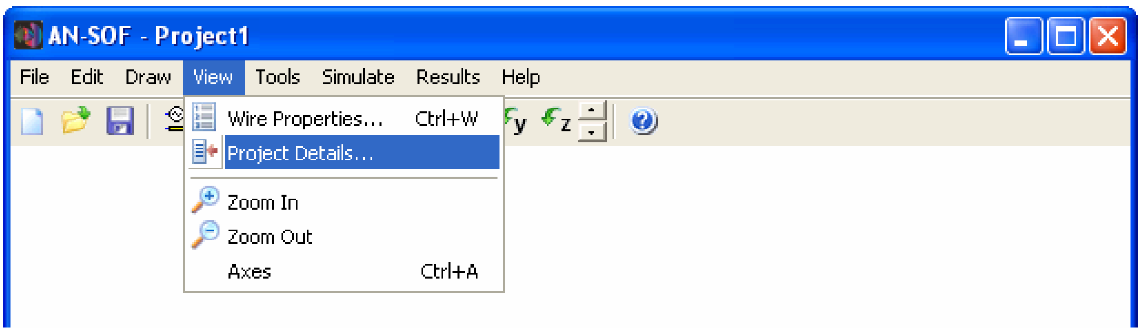


Fig. 6.14: Project Details... command in the main menu

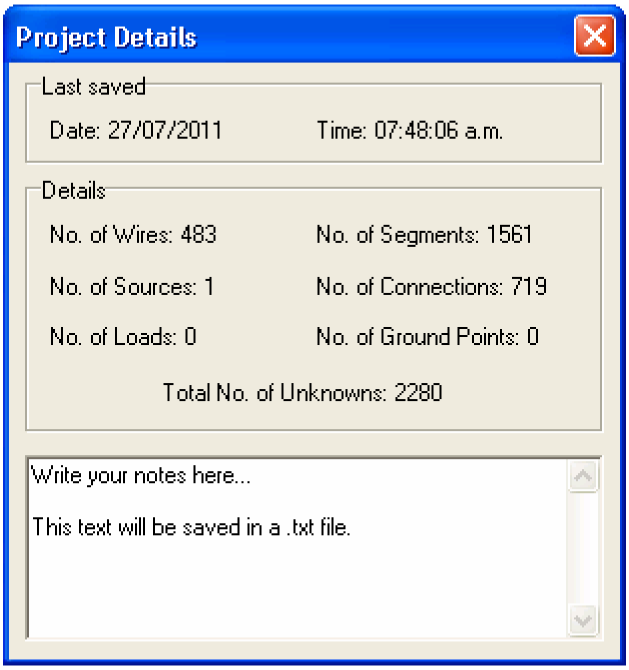


Fig. 6.15: Project Details dialog box.

6.10 Tapered Wires

A tapered wire is a wire with a variable radius along its length, so the cross section of tapered wires is always circular. The radius is varied linearly along the wire and in defined steps, then a wire with a stepped radius is obtained, as shown in Fig. 6.16.

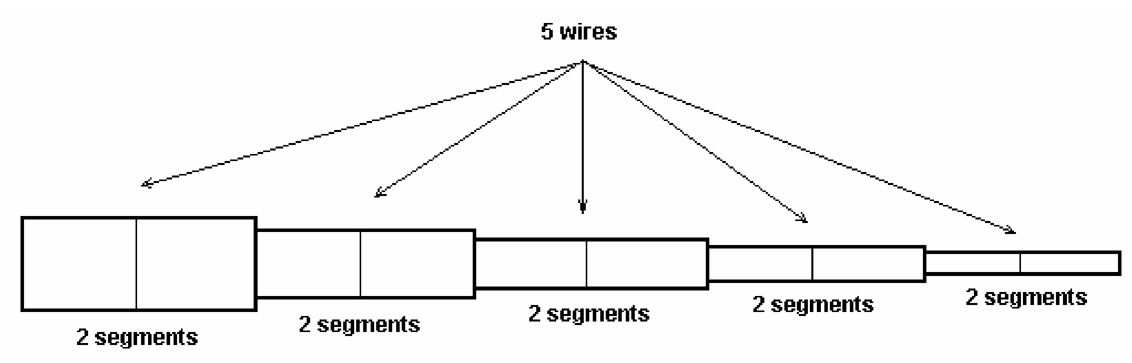


Fig. 6.16: Example of a tapered wire divided into 5 wire portions. Each wire portion is divided into 2 segments.

Choose Draw/Tapered Wire in the main menu and select a wire type for drawing, Fig. 6.17. The wire types available are the same as in the Draw/Wire menu. As an example, Fig. 6.18 shows the Line page of the Draw dialog box when a line wire is selected.

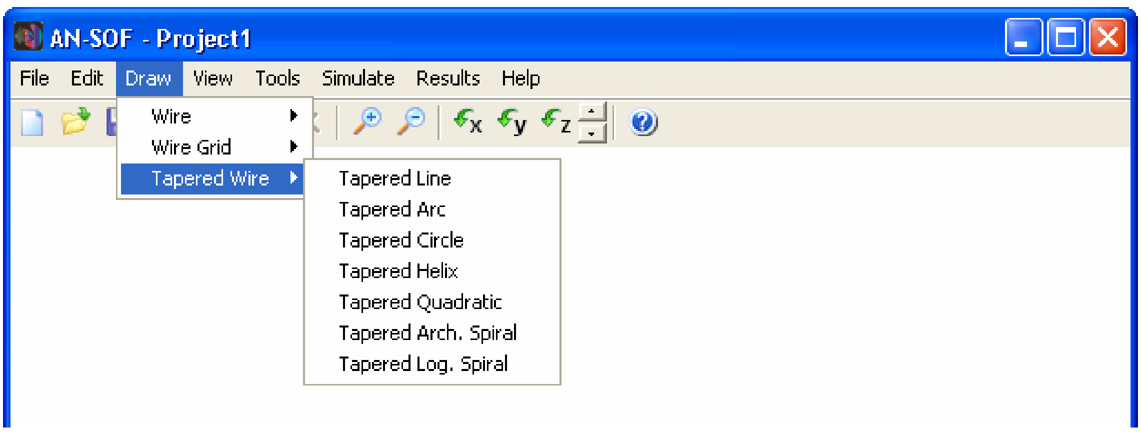


Fig. 6.17: Draw/Tapered Wire option in the main menu.

The wire must be divided into wire portions according to the desired steps in radius, as it is indicated in Fig. 6.16. Also, each wire portion having a uniform radius must be divided into segments as it is required by the Method of Moments used for the simulation.

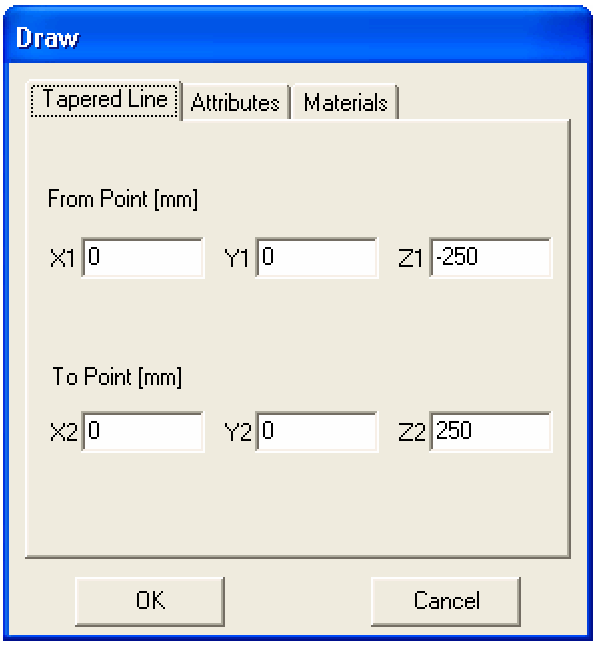


Fig. 6.18: Tapered Line page in the Draw dialog box when the Draw/Tapered Wire/Tapered Line option is chosen in the main menu.

The number of wire portions and the number of segments per wire can be set by choosing the Attributes page, Fig. 6.19. In this page, the Start and End radii can be set.

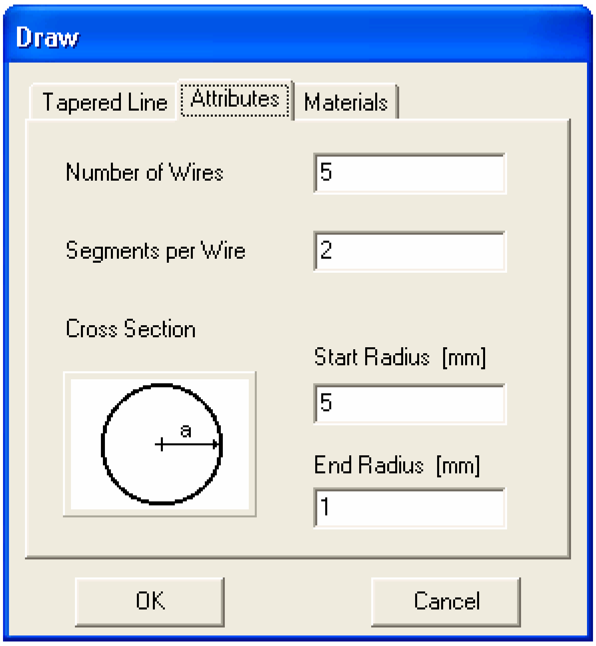


Fig. 6.19: Attributes page where the number of wire portions and segments per wire can be set, as well as Start and End radii.

The resistivity for the conductive wire and its coating material can be set in the Materials page, Fig. 6.20. In this case, a tapered coating shield can also be defined by giving an Start and End thickness.

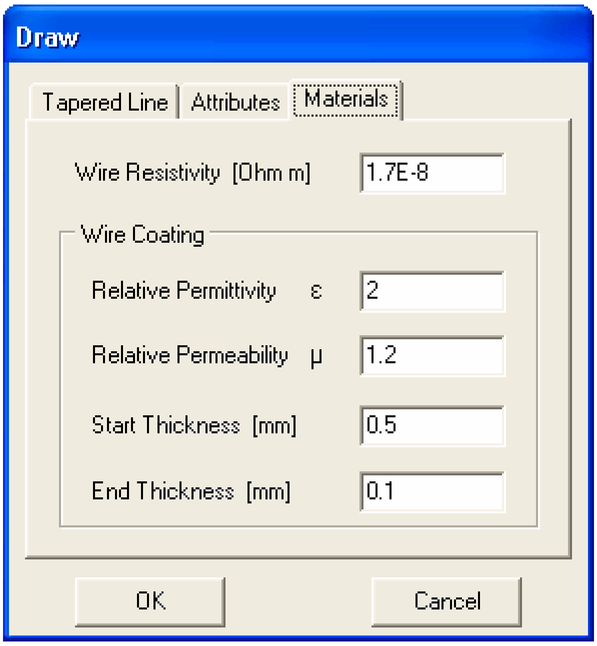


Fig. 6.20: Materials page where the wire resistivity and coating can be set. A tapered coating can be defined by giving the Start and End thicknesses.

7. Wire Grids

Wire grids can be composed of curved or straight wire segments and can be used to model grids and approximate conductive surfaces.

AN-SOF® has different types of wire grids. Any wire grid type has its own geometrical parameters and attributes that can be set in its specific Draw dialog box. This dialog box allows the user creating and drawing a new wire grid on the project workspace.

Choosing Draw/Wire Grid in the main menu shows a sub-menu with the following commands, Fig. 7.1:

- **Plate:** Displays the Draw dialog box for drawing a plate or bilinear surface.
- **Disc:** Displays the Draw dialog box for drawing a disc.
- **Flat Ring:** Displays the Draw dialog box for drawing a flat ring or a disc with a hole at its center.
- **Cone:** Displays the Draw dialog box for drawing a cone.
- **Truncated Cone:** Displays the Draw dialog box for drawing a truncated cone.
- **Cylinder:** Displays the Draw dialog box for drawing a cylinder.
- **Sphere:** Displays the Draw dialog box for drawing a sphere.
- **Paraboloid:** Displays the Draw dialog box for drawing a paraboloid.

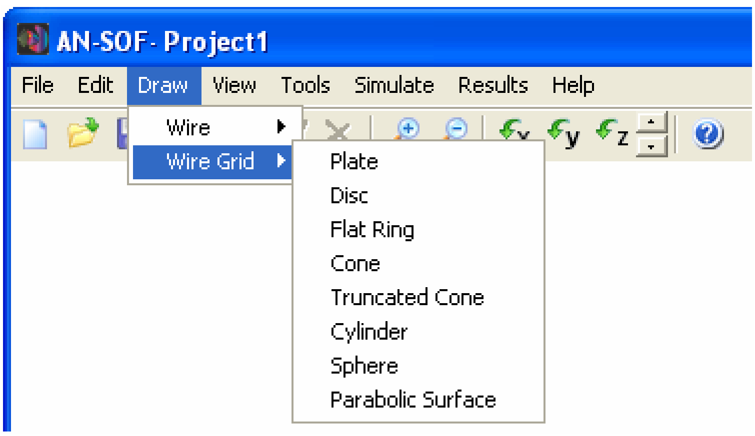


Fig. 7.1: Draw/Wire Grid option in the main menu.

7.1 Plate

The Plate refers to a plate or bilinear surface.

Choose Draw/Wire Grid/Plate in the main menu to display the Draw dialog box for the Plate, Fig 7.2. This dialog box has two pages: Plate and Attributes, Fig. 7.3.

The Plate page

The Plate page sets the geometrical parameters for the Plate.

The Plate is defined by giving the coordinates of four corner points.
In the general case, a plate or bilinear surface is a non-planar quadrilateral, which is defined uniquely by its four vertices, as shown in Fig. 7.4. In the particular case, a bilinear surface degenerates into a flat quadrilateral, rectangle, triangle, square, etc.

Once the geometrical parameters in the Plate page have been set, the Attributes page can be chosen, where the number of facets of the Plate can be entered. Section 7.9 describes other parameters that can be defined in the Attributes page.

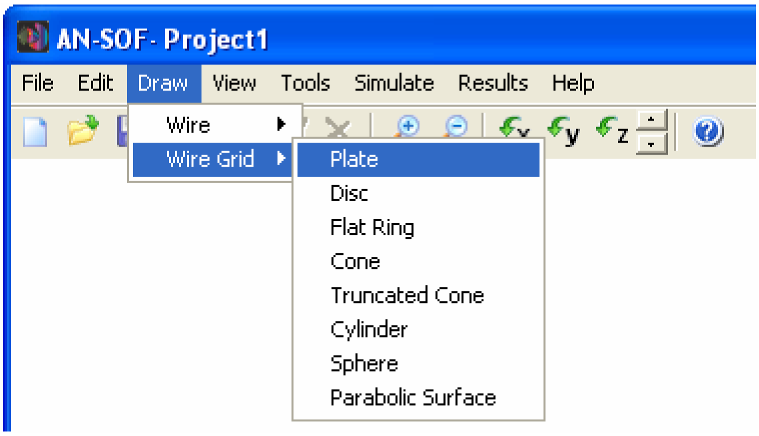


Fig. 7.2: The Draw/Wire Grid/Plate command in the main menu displays the Draw dialog box for the Plate.

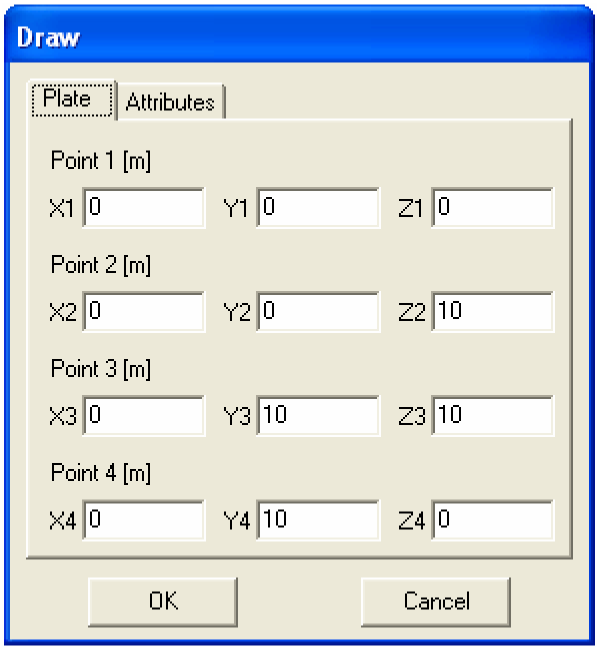


Fig. 7.3: Plate page of the Draw dialog box.

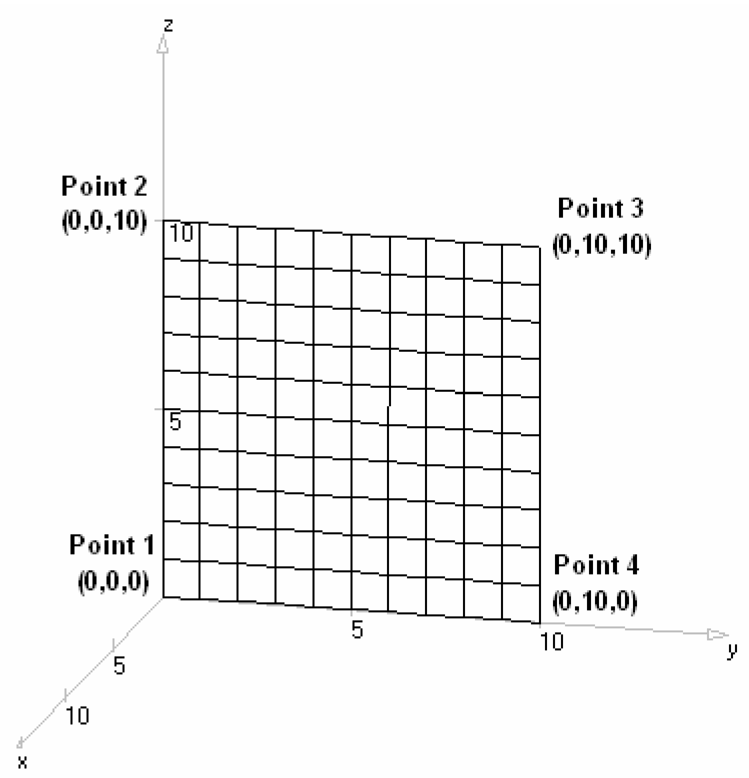


Fig. 7.4: A Plate drawn using the input data of Fig. 7.3.

7.2 Disc

The Disc refers to a disc or circular surface.

Choose Draw/Wire Grid/Disc in the main menu to display the Draw dialog box for the Disc, Fig 7.5. This dialog box has two pages: Disc and Attributes, Fig. 7.6.

The Disc page

The Disc page sets the geometrical parameters for the Disc. There is a combo-box with two options: Curved segments and Straight segments. Choose Curved segments for an exact representation of the disc curvature. The Straight segments option is the typical approximation using straight or linear wires.

The Disc is defined by giving the Center coordinates, Radius and orientation angles, Theta and Phi. A disc is a planar surface, which is defined uniquely by these parameters, as shown in Fig. 7.7.

Once the geometrical parameters in the Disc page have been set, the Attributes page can be chosen, where the number of facets of the Disc can be entered. Section 7.9 describes other parameters that can be defined in the Attributes page.

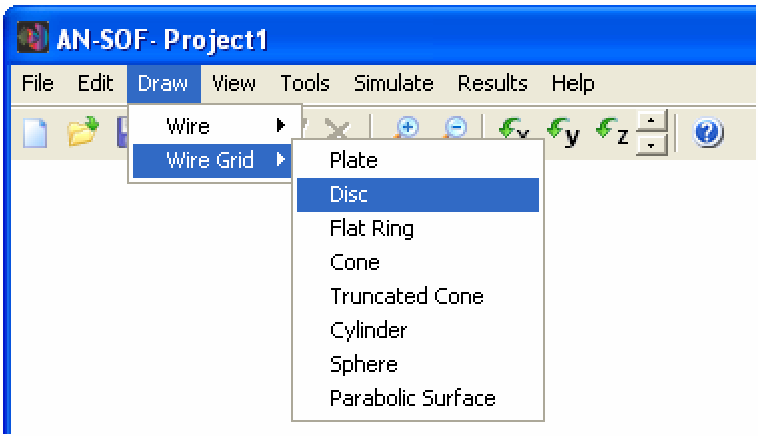


Fig. 7.5: The Draw/Wire Grid/Disc command in the main menu displays the Draw dialog box for the Disc.

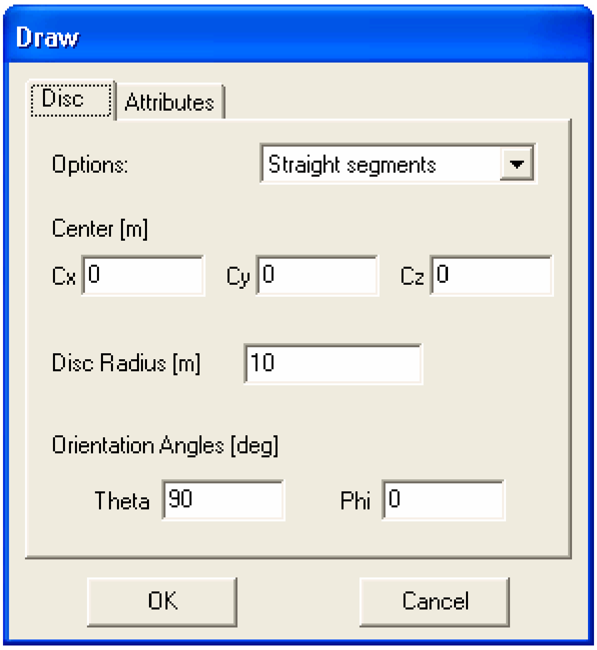


Fig. 7.6: Disc page of the Draw dialog box.

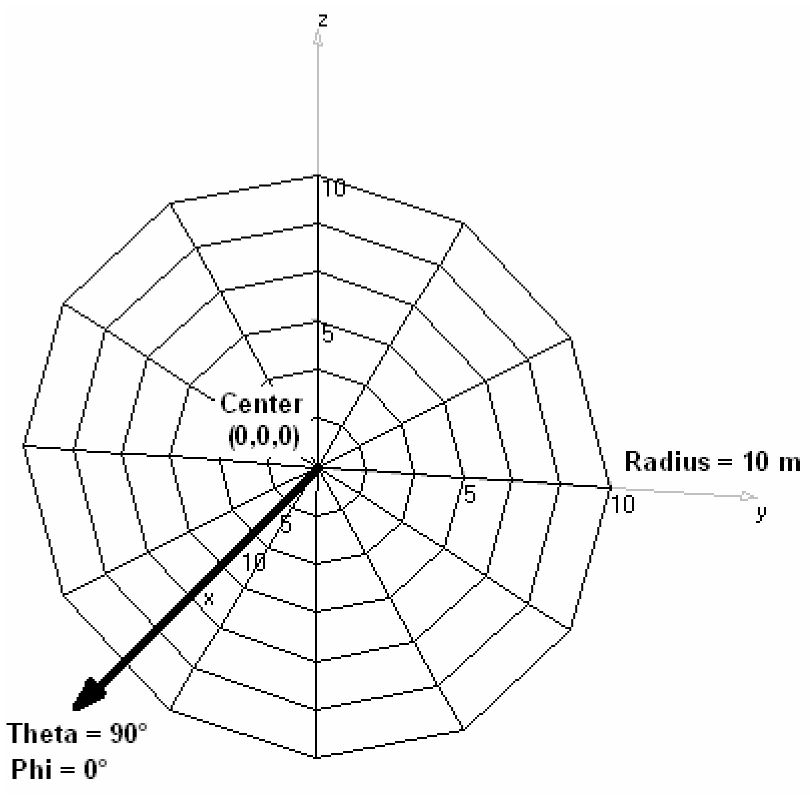


Fig. 7.7: A Disc drawn using the input data of Fig. 7.6.

7.3 Flat Ring

The Flat Ring refers to a disc with a hole at its center.

Choose Draw/Wire Grid/Flat Ring in the main menu to display the Draw dialog box for the Flat Ring, Fig 7.8. This dialog box has two pages: Flat Ring and Attributes, Fig. 7.9.

The Flat Ring page

The Flat Ring page sets the geometrical parameters for the Flat Ring. There is a combo-box with two options: Curved segments and Straight segments. Choose Curved segments for an exact representation of the flat ring curvature. The Straight segments option is the typical approximation using straight or linear wires.

The Flat Ring is defined by giving the Center coordinates, Inner Radius (hole radius), Outer Radius and orientation angles, Theta and Phi. A flat ring is a planar surface, which is defined uniquely by these parameters, as shown in Fig. 7.10.

Once the geometrical parameters in the Flat Ring page have been set, the Attributes page can be chosen, where the number of facets of the Flat Ring can be entered. Section 7.9 describes other parameters that can be defined in the Attributes page.

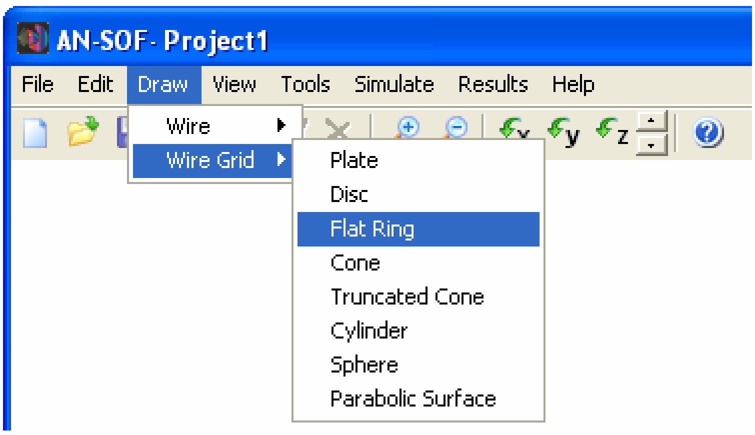


Fig. 7.8: The Draw/Wire Grid/Flat Ring command in the main menu displays the Draw dialog box for the Flat Ring.

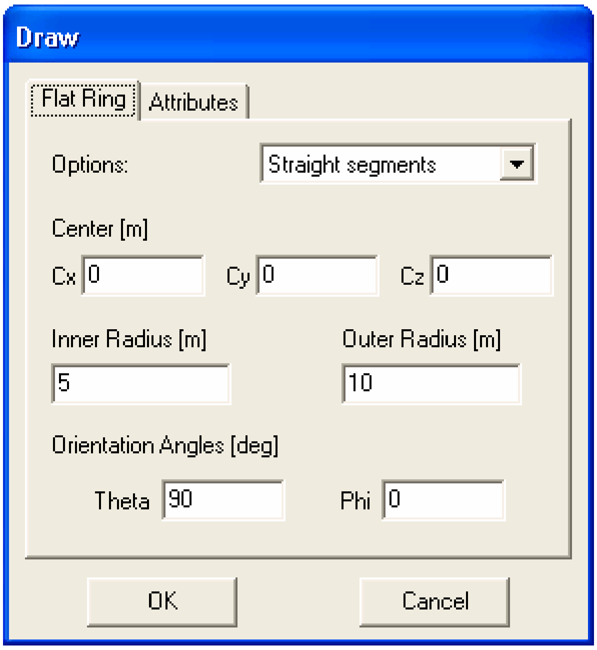


Fig. 7.9: Flat Ring page of the Draw dialog box.

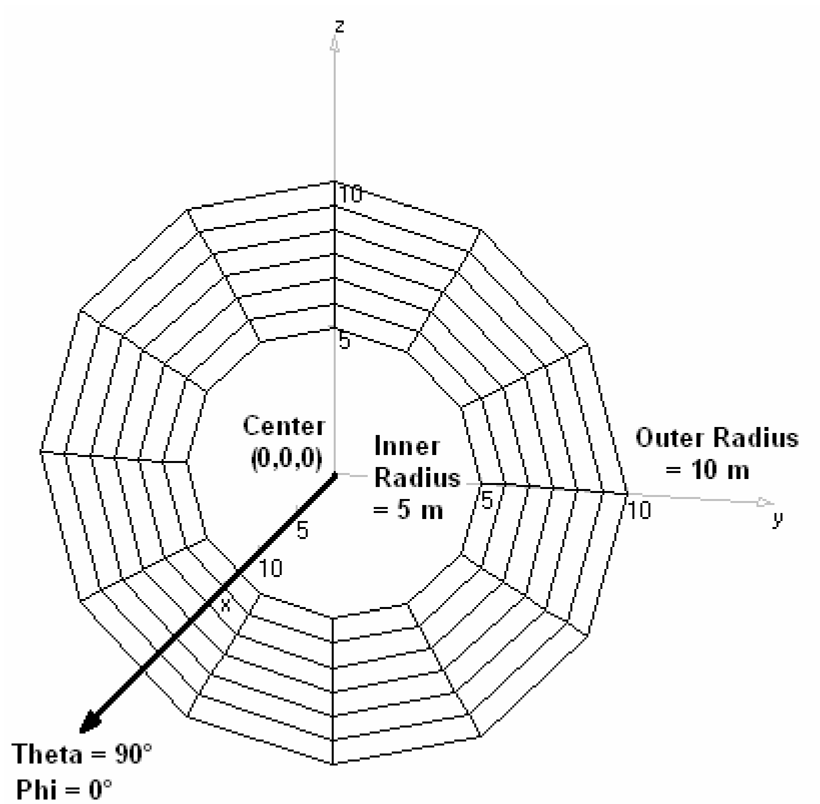


Fig. 7.10: A Flat Ring drawn using the input data of Fig. 7.9.

7.4 Cone

Choose Draw/Wire Grid/Cone in the main menu to display the Draw dialog box for the Cone, Fig 7.11. This dialog box has two pages: Cone and Attributes, Fig. 7.12.

The Cone page

The Cone page sets the geometrical parameters for the Cone. There is a combo-box with two options: Curved segments and Straight segments. Choose Curved segments for an exact representation of the cone curvature. The Straight segments option is the typical approximation using straight or linear wires.

The Cone is defined by giving the Vertex coordinates, Aperture Angle, Aperture Radius and orientation angles, Theta and Phi. A cone is a surface which is defined uniquely by these parameters, as shown in Fig. 7.13.

Once the geometrical parameters in the Cone page have been set, the Attributes page can be chosen, where the number of facets of the Cone can be entered. Section 7.9 describes other parameters that can be defined in the Attributes page.

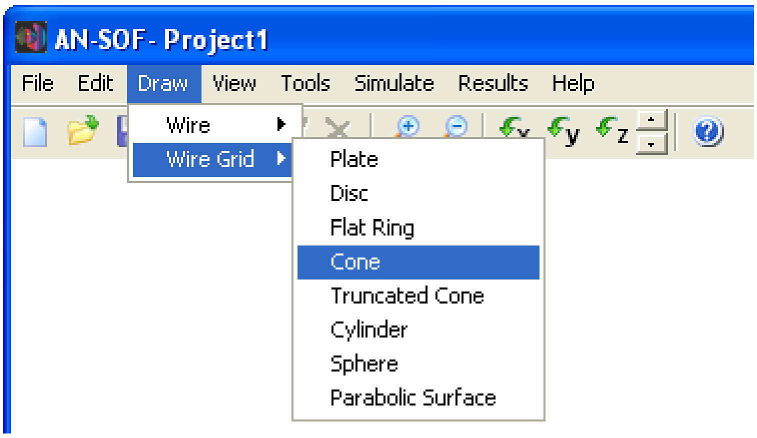


Fig. 7.11: The Draw/Wire Grid/Cone command in the main menu displays the Draw dialog box for the Cone.

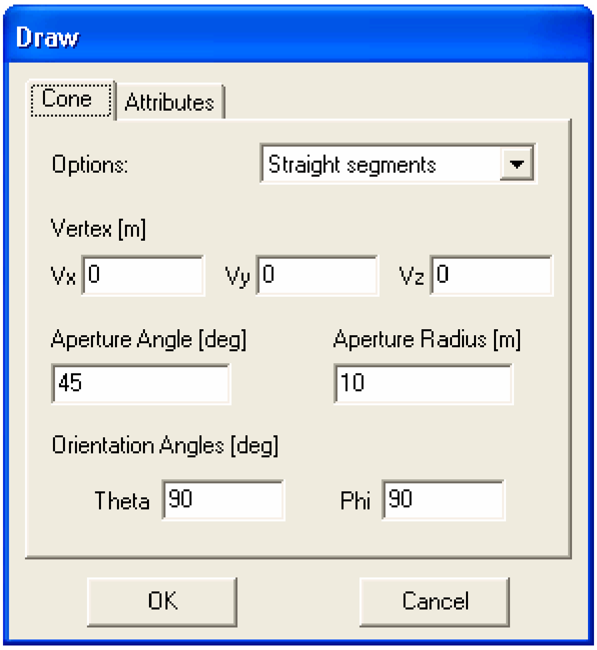


Fig. 7.12: Cone page of the Draw dialog box.

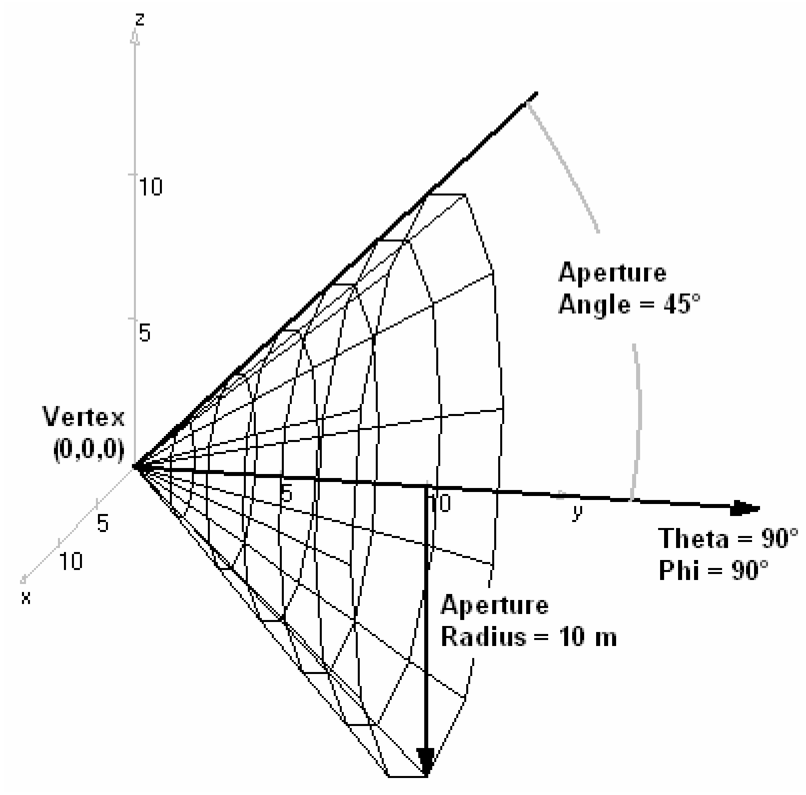


Fig. 7.13: A Cone drawn using the input data of Fig. 7.12.

7.5 Truncated Cone

Choose Draw/Wire Grid/Truncated Cone in the main menu to display the Draw dialog box for the Truncated Cone, Fig 7.14. This dialog box has two pages: Truncated Cone and Attributes, Fig. 7.15.

The Truncated Cone page

The Truncated Cone page sets the geometrical parameters for the Truncated Cone. There is a combo-box with two options: Curved segments and Straight segments. Choose Curved segments for an exact representation of the truncated cone curvature. The Straight segments option is the typical approximation using straight or linear wires.

The Truncated Cone is defined by giving the Base Point coordinates, Base Radius, Top Radius, Aperture angle and orientation angles, Theta and Phi. A truncated cone is a surface which is defined uniquely by these parameters, as shown in Fig. 7.16. A truncated cone can degenerate into a cylinder, a cone, a disc and a flat ring.

Once the geometrical parameters in the Truncated Cone page have been set, the Attributes page can be chosen, where the number of facets of the Truncated Cone can be entered. Section 7.9 describes other parameters that can be defined in the Attributes page.

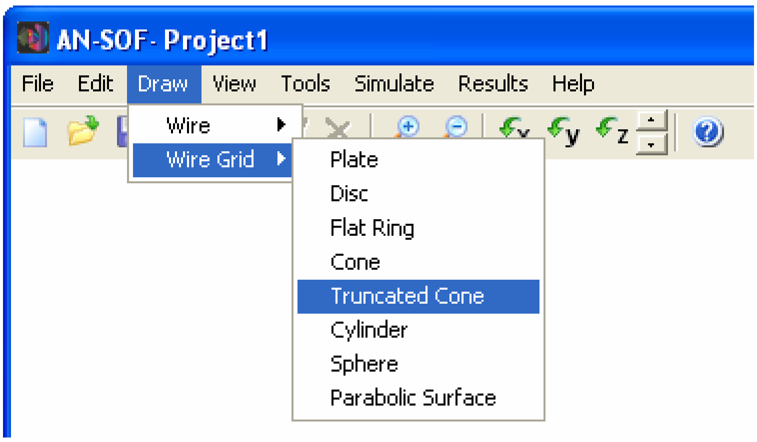


Fig. 7.14: The Draw/Wire Grid/Truncated Cone command in the main menu displays the Draw dialog box for the Truncated Cone.

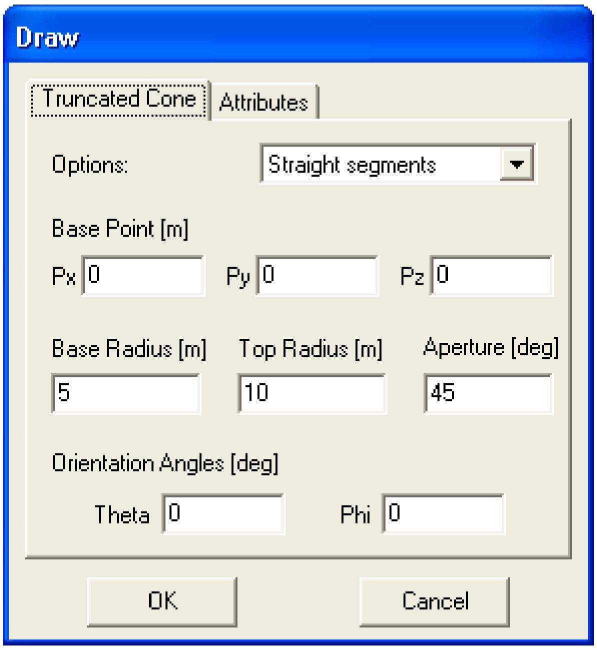


Fig. 7.15: Truncated Cone page of the Draw dialog box.

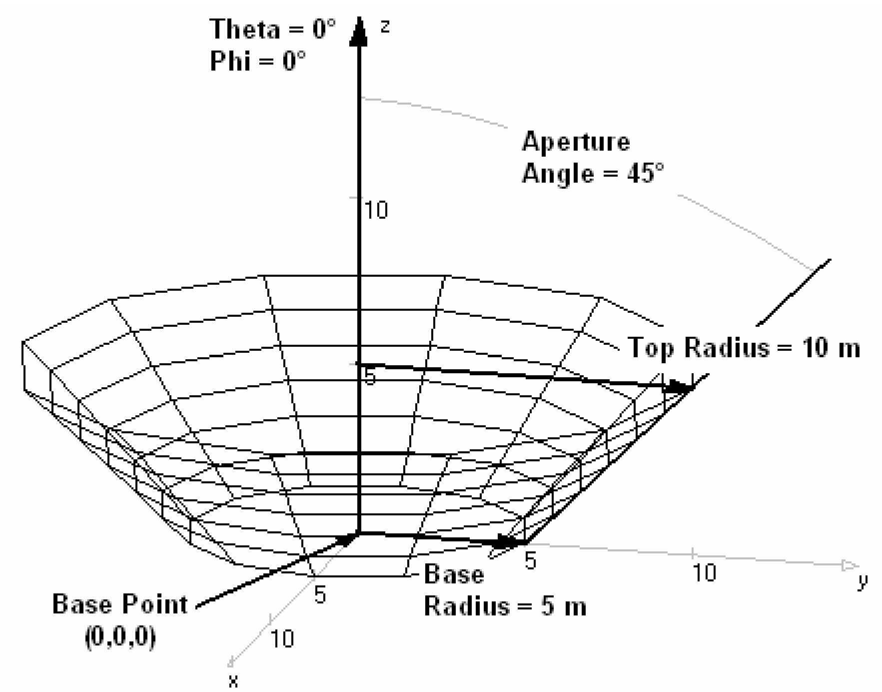


Fig. 7.16: A Truncated Cone drawn using the input data of Fig. 7.15.

7.6 Cylinder

Choose Draw/Wire Grid/Cylinder in the main menu to display the Draw dialog box for the Cylinder, Fig 7.17. This dialog box has two pages: Cylinder and Attributes, Fig. 7.18.

The Cylinder page

The Cylinder page sets the geometrical parameters for the Cylinder. There is a combo-box with two options: Curved segments and Straight segments. Choose Curved segments for an exact representation of the cylinder curvature. The Straight segments option is the typical approximation using straight or linear wires.

The Cylinder is defined by giving the Base Point coordinates, Length, Radius and orientation angles, Theta and Phi. A cylinder is a surface which is defined uniquely by these parameters, as shown in Fig. 7.19.

Once the geometrical parameters in the Cylinder page have been set, the Attributes page can be chosen, where the number of facets of the Cylinder can be entered. Section 7.9 describes other parameters that can be defined in the Attributes page.

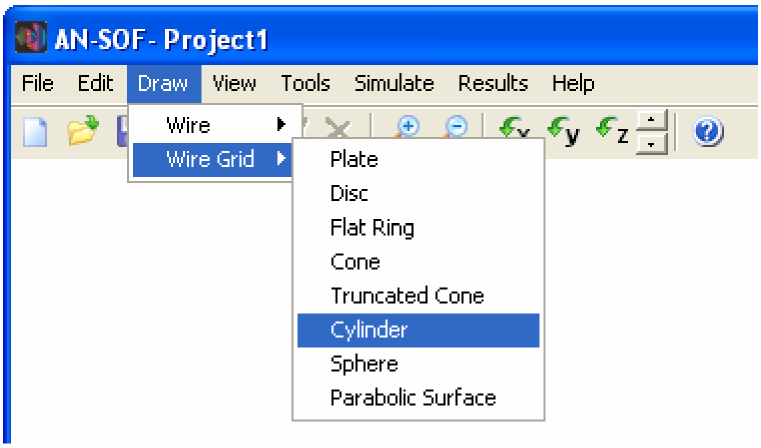


Fig. 7.17: The Draw/Wire Grid/Cylinder command in the main menu displays the Draw dialog box for the Cylinder.

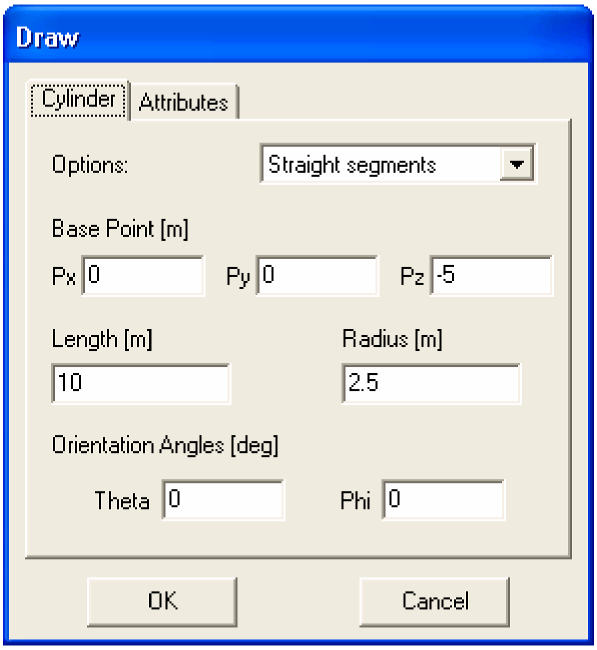


Fig. 7.18: Cylinder page of the Draw dialog box.

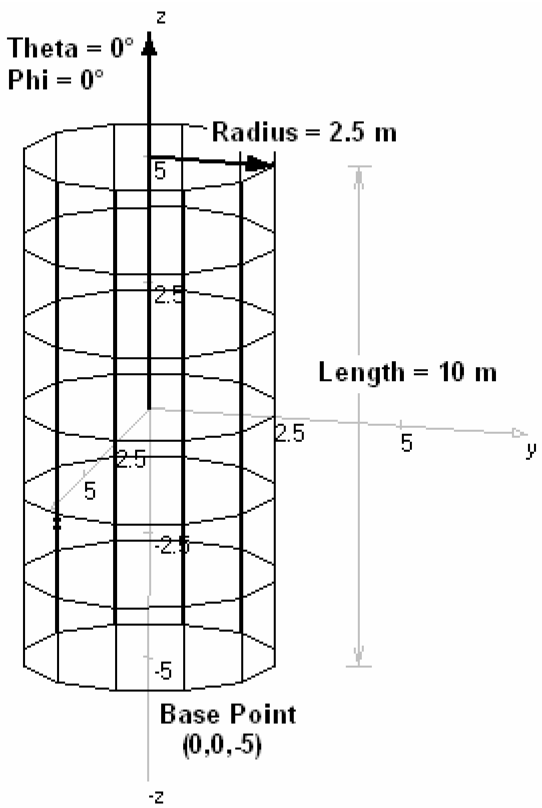


Fig. 7.19: A Cylinder drawn using the input data of Fig. 7.18.

7.7 Sphere

Choose Draw/Wire Grid/Sphere in the main menu to display the Draw dialog box for the Sphere, Fig 7.20. This dialog box has two pages: Sphere and Attributes, Fig. 7.21.

The Sphere page

The Sphere page sets the geometrical parameters for the Sphere. There is a combo-box with two options: Curved segments and Straight segments. Choose Curved segments for an exact representation of the sphere curvature. The Straight segments option is the typical approximation using straight or linear wires.

The Sphere is defined by giving the Center coordinates, Radius and orientation angles, Theta and Phi. A sphere is a surface which is defined uniquely by these parameters, as shown in Fig. 7.22.

Once the geometrical parameters in the Sphere page have been set, the Attributes page can be chosen, where the number of facets of the Sphere can be entered. Section 7.9 describes other parameters that can be defined in the Attributes page.

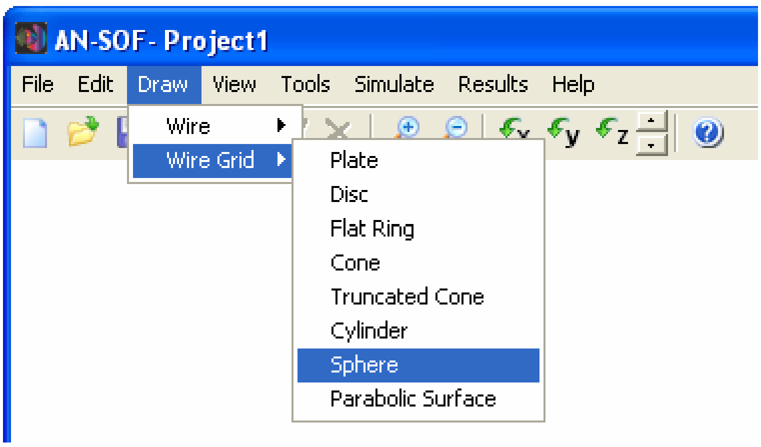


Fig. 7.20: The Draw/Wire Grid/Sphere command in the main menu displays the Draw dialog box for the Sphere.

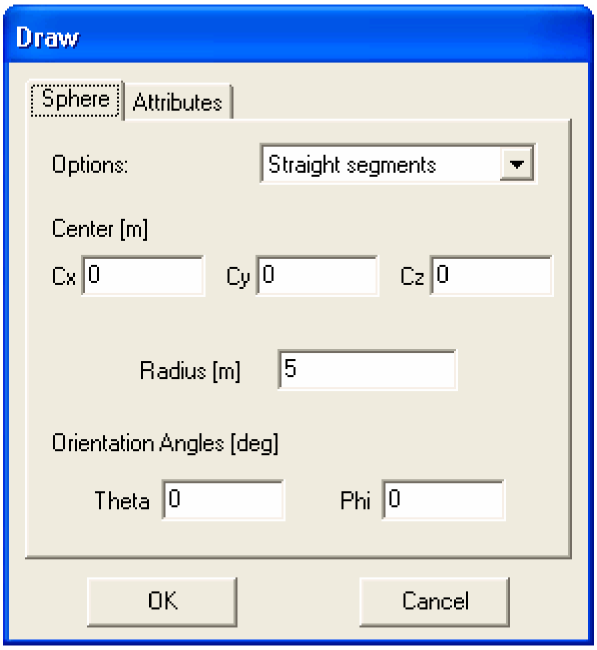


Fig. 7.21: Sphere page of the Draw dialog box.

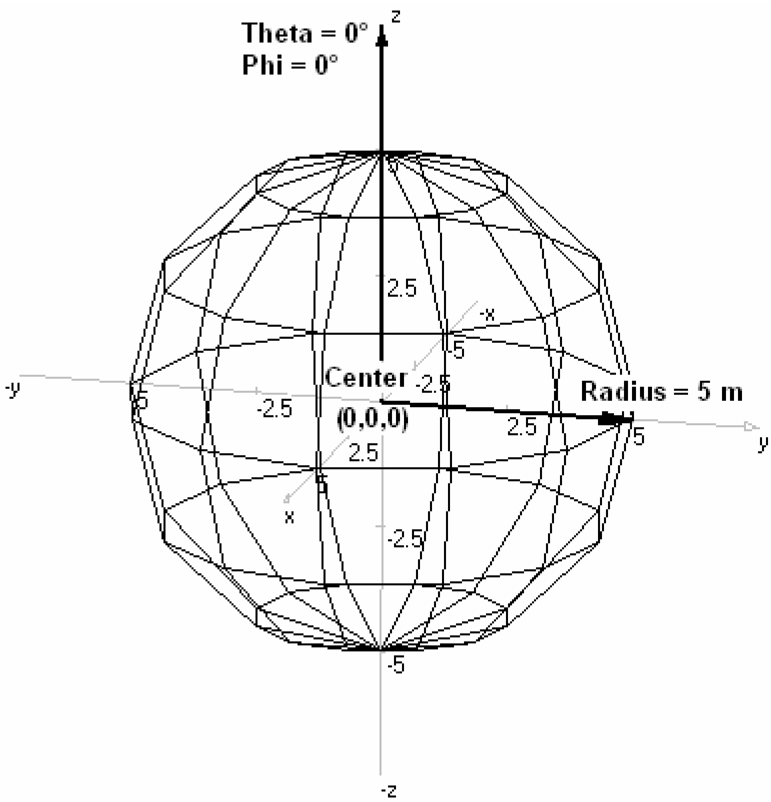


Fig. 7.22: A Sphere drawn using the input data of Fig. 7.21.

7.8 Paraboloid

Choose Draw/Wire Grid/Paraboloid in the main menu to display the Draw dialog box for the Paraboloid, Fig 7.23. This dialog box has two pages: Paraboloid and Attributes, Fig. 7.24.

The Paraboloid page

The Paraboloid page sets the geometrical parameters for the Paraboloid. There is a combo-box with two options: Curved segments and Straight segments. Choose Curved segments for an exact representation of the paraboloid curvature. The Straight segments option is the typical approximation using straight or linear wires.

The Paraboloid is defined by giving the Vertex coordinates, Focal Distance, Aperture Radius and orientation angles, Theta and Phi. A paraboloid or parabolic surface is a curved surface which is defined uniquely by these parameters, as shown in Fig. 7.25.

Once the geometrical parameters in the Paraboloid page have been set, the Attributes page can be chosen, where the number of facets of the Paraboloid can be entered. Section 7.9 describes other parameters that can be defined in the Attributes page.

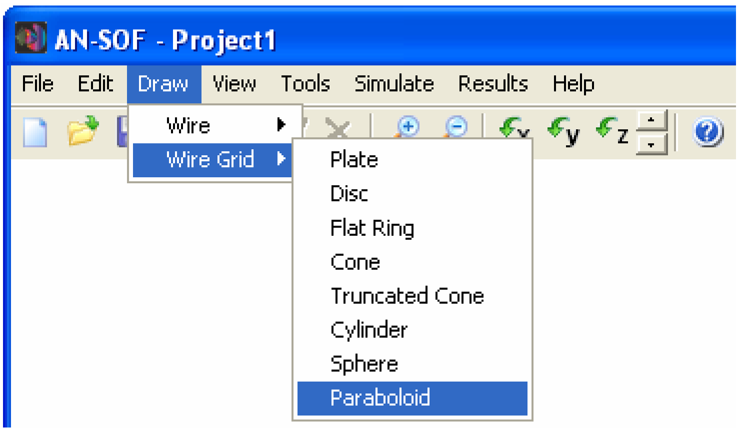


Fig. 7.23: The Draw/Wire Grid/Paraboloid command in the main menu displays the Draw dialog box for the Paraboloid.

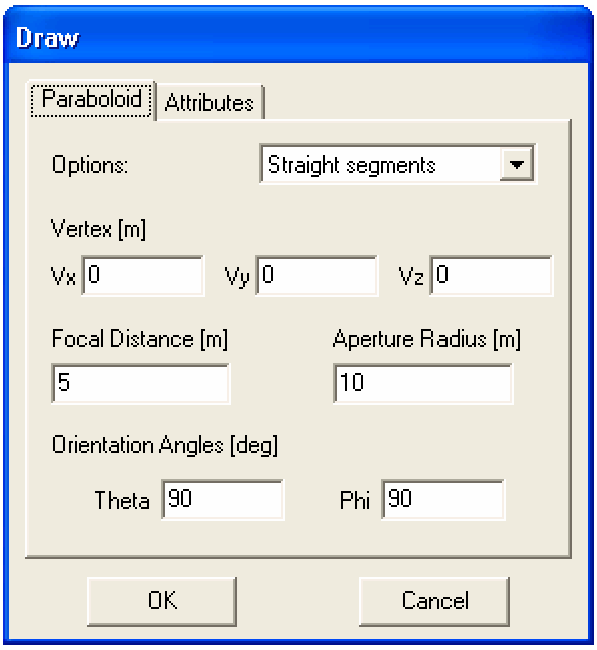


Fig. 7.24: Paraboloid page of the Draw dialog box.

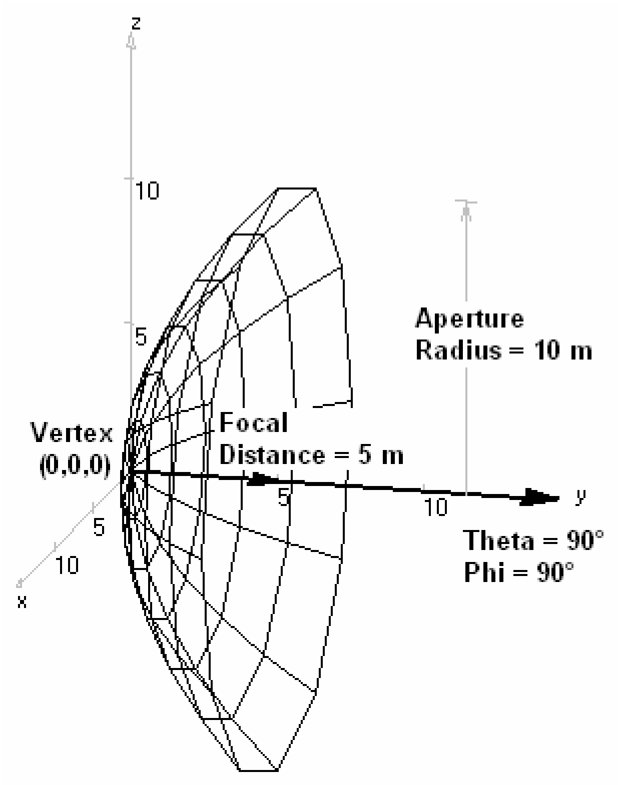


Fig. 7.25: A Paraboloid drawn using the input data of Fig. 7.24.

7.9 Wire Grid Attributes

The Attributes page belongs to the Draw dialog box of the chosen wire grid type, Fig. 7.26. In the Attributes page the following attributes can be specified:

Number of facets

Any wire grid has a given number of facets. For example, the plate in Fig. 7.4 has 10x10 facets and the disc in Fig 7.7 has 6x12 facets.
Any facet is a quadrilateral composed of four wires, which are divided into segments according to the shortest wavelength or highest frequency. An unknown current on each wire segment must be found in the simulation process. Any curved or straight wire composing a wire grid can be edited individually, following the procedures explained in Chapters 5 and 6.

Resistivity [Ohm m]

A non-zero finite resistivity, in [Ohms meter], for the wires in a wire grid can be specified. This value is used for computing a distributed resistance along the wires, taking into account the skin effect.
The resistivity equals the inverse of the conductivity in [S/m] (Siemens per meter). For copper wires, the conductivity is 5.8×10^{7} S/m = 5.8E7 S/m, then the resistivity will be $(5.8 \times 10^{7}\ \mathrm{S/m})^{-1} = 1.72 \times 10^{-8}$ Ohm m = 1.72E-8 Ohm m.

The resistivity of wires is taken into account in the computation if the option Resistivity is checked in the Options page of the Configuration dialog box.

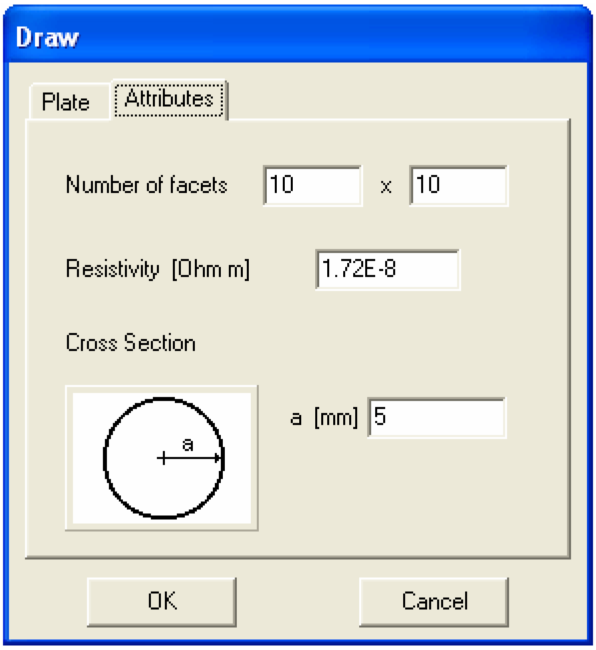


Fig. 7.26: Attributes page in the Draw dialog box for the Plate wire grid.

Cross-Section

The cross-section of the wires in a wire grid is circular. Infinitesimally thin wires are not allowed, so the cross-section radius “a” must be greater than zero.

The Draw dialog box for any wire grid type has its own Attributes page with the same features described here.

7.10 Deleting a Wire Grid

Clicking the left mouse button on the workspace, while maintaining pressed the Ctrl key, a selecting box can be expanded, Fig. 7.27. This selecting box permits selecting a wire grid or any group of wires in the project workspace, which is highlighted in red.

Choosing the Edit/Delete command in the main menu deletes the selected wire grid.

The Delete command can also be executed by pressing Del on the keyboard or the Edit toolbar.

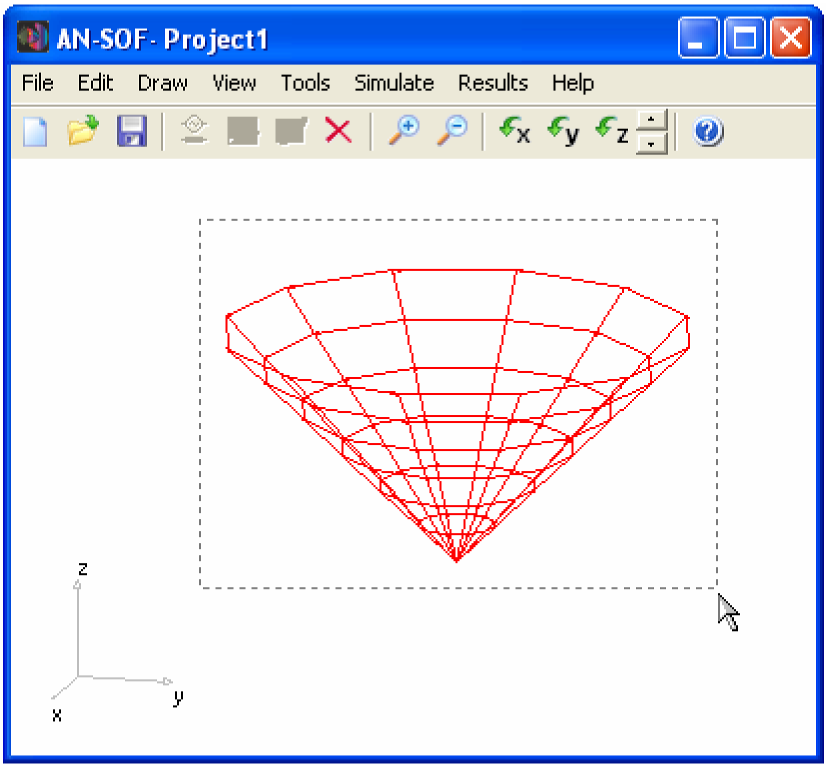


Fig. 7.27: Box for selecting a wire grid. The left mouse button and Ctrl key must be pressed at the same time.

7.11 Wire Grid Color

Clicking the left mouse button on the workspace, while maintaining pressed the Ctrl key, a selecting box can be expanded, Fig. 7.27. This selecting box permits selecting a wire grid or any group of wires in the project workspace, which is highlighted in red.

Choosing the Edit/Wire Color command in the main menu shows a Windows dialog box for selecting a color for the group of wires.

The Wire Color command can also be executed by pressing the Wire Color button in the Edit toolbar.

8. Sources and Loads

An arbitrary number of discrete sources can be placed at any positions for the excitation of the structure. There are two types of sources:

- ✓ **Voltage sources**
- ✓ **Current sources**

Current sources can be used for the simulation of impressed currents.

Their amplitudes and phases define discrete sources, which can also have internal impedances. These internal impedances can either be resistive, inductive or capacitive.

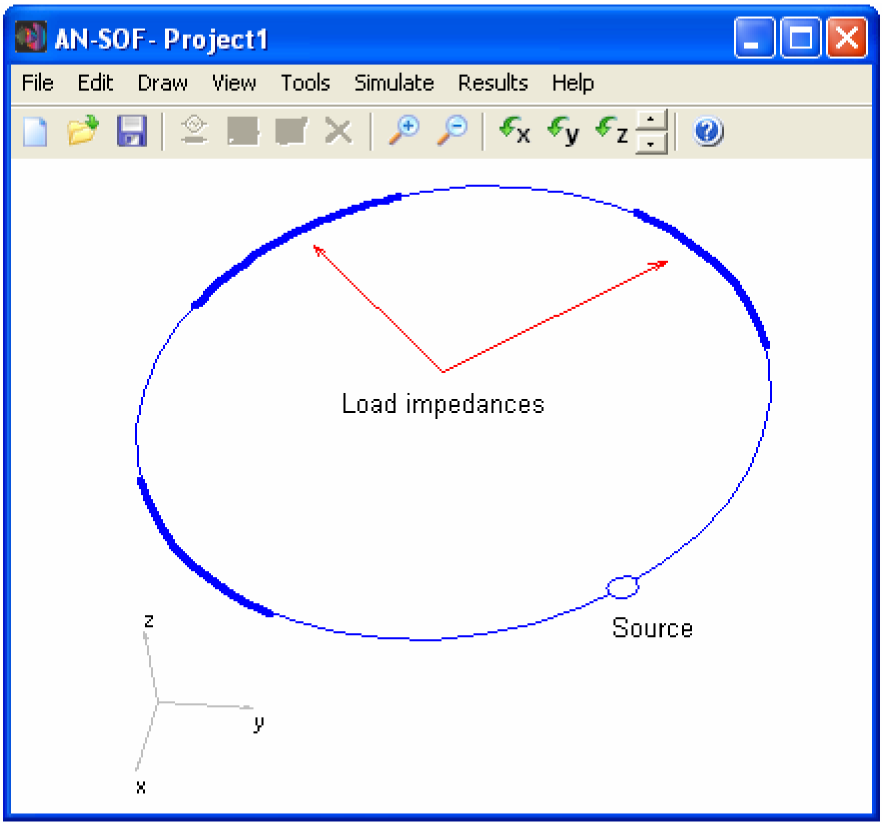


Fig. 8.1: 3D symbols used by the program for showing the source and load positions.

Lumped loads, representing discrete resistors, inductors and capacitors can be added to a wire at any position. There are two types of loads:

- ✓ **Inductive**
- ✓ **Capacitive**

The inductive load impedance is defined by giving a resistance in [Ohm] and an inductance. An inductive load with a null inductance will give a pure resistor. Also, a capacitive load impedance is defined by giving a resistance in [Ohm] and a capacitance. The inductance unit can either be pH, nH, uH, mH or H, while the capacitance unit can either be pF, nF, uF, mF or F. These units are defined in the Preferences dialog box.

The source and load positions are shown on the workspace with special 3D-symbols, Fig. 8.1.

8.1 Choosing Sources as the Excitation

When discrete sources have to be used as the excitation for the wire structure, the **Incident Plane Wave** option in the Options page of the Configuration dialog box must be **unchecked**.

1. Choose Simulate/Configure... in the main menu to display the Configuration dialog box.
2. Select the Options page. The Incident Plane Wave option is in this page.
3. The Incident Plane Wave option must be unchecked, as shown in Fig. 8.2.

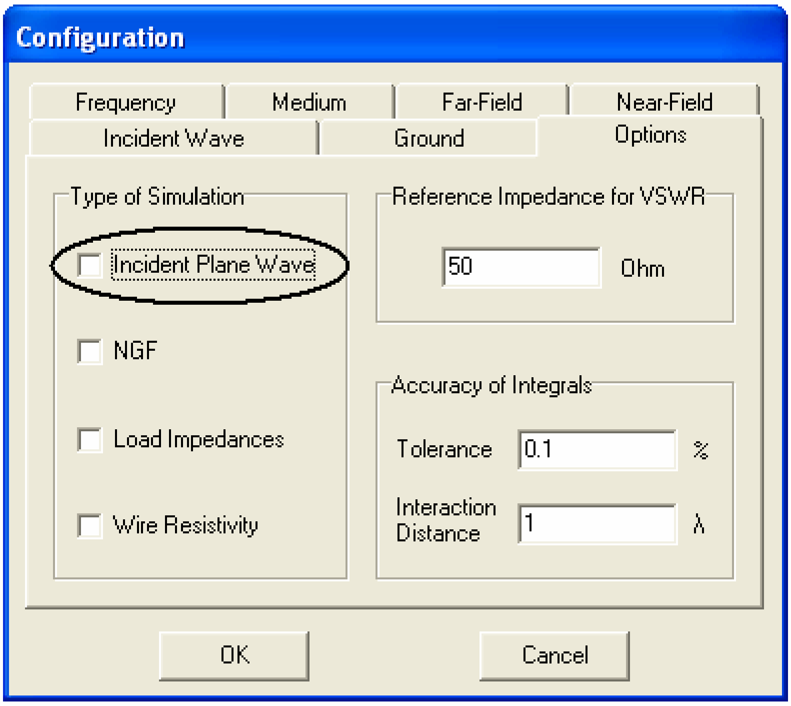


Fig. 8.2: The Incident Plane Wave option in the Options page of the Configuration dialog box must be unchecked when sources have to be used as the excitation.

8.2 The Source/Load Toolbar

Discrete sources and loads can be placed at any positions on a selected wire by using the Source/Load toolbar. Sources an loads can also be modified by means of this toolbar.

By clicking with the right mouse button in any part of a wire a pop-up menu will be shown, Fig. 8.3. Choose the Source/Load command from the pop-up menu to display the Source/Load toolbar, Fig. 8.5.

The Source/Load command can also be chosen by first selecting a wire by clicking the left mouse button on it, and next choosing Edit/Source/Load in the main menu, Fig. 8.4.

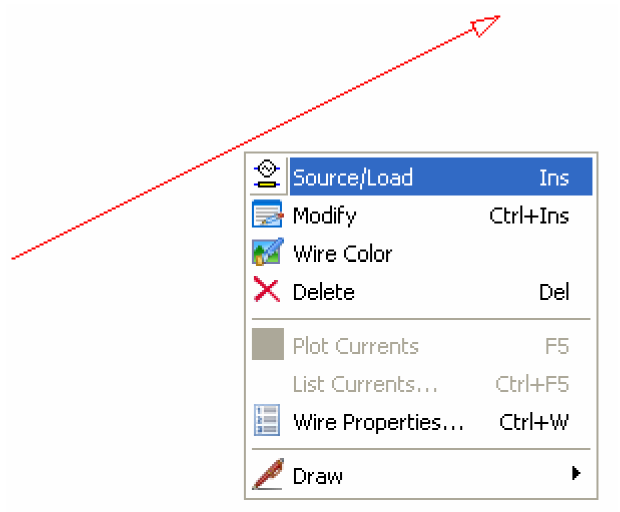


Fig. 8.3: Source/Load command in the pop-up menu.

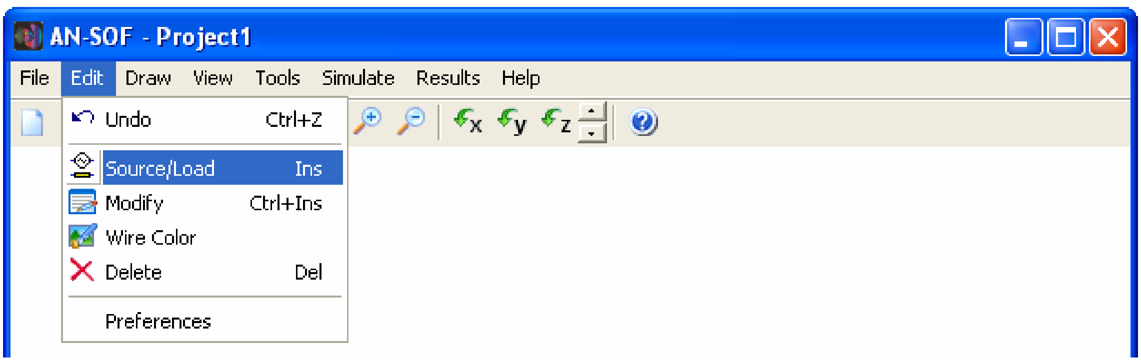


Fig. 8.4: Source/Load command in the main menu.

The Source/Load toolbar has the following components:

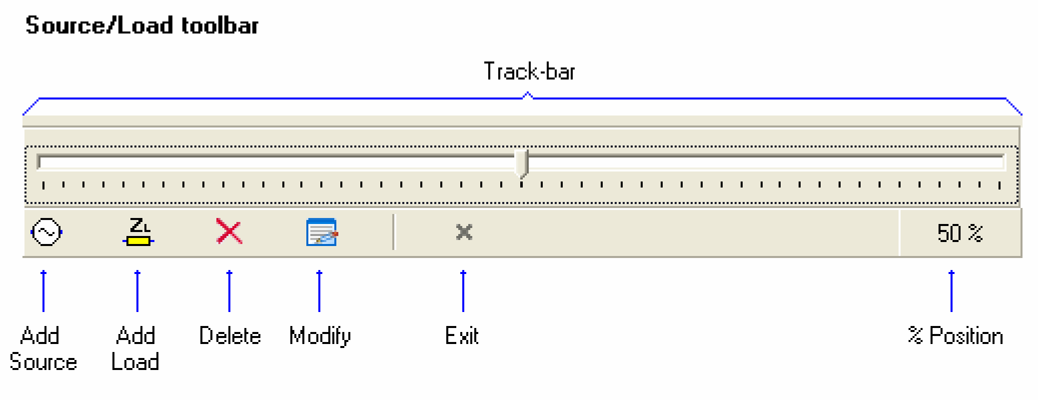


Fig. 8.5: Source/Load toolbar.

The Track-bar

Each position of the Track-bar corresponds to the position of a segment in the selected wire. Thus, the Track-bar allows us selecting a particular segment on the wire. At the right corner of the Track-bar the position of the selected segment is is shown as a percentage of the wire length. The segment position is measured from the starting point of the wire to the middle point of the selected segment, and it is defined as follows:

$$\% \text{ position} = 100 \text{ (position / wire length)}$$

The Add Source button

Displays the Add Source dialog box for adding a source to the selected segment in the wire, Fig. 8.6. This dialog box allows us choosing the type of source, its amplitude, phase and internal impedance.

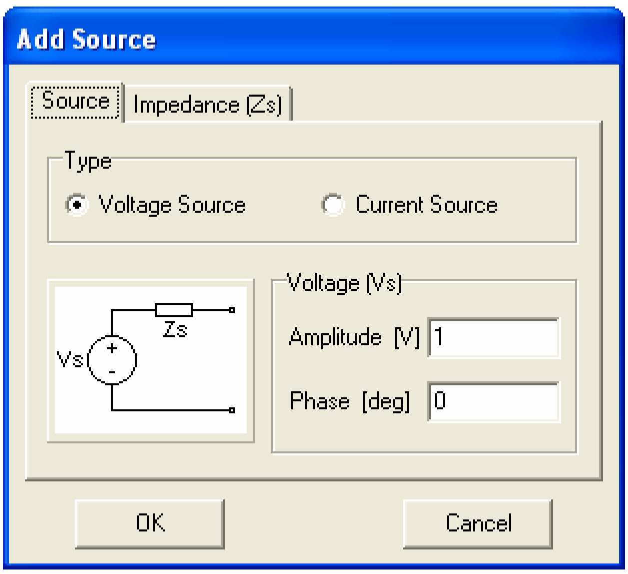


Fig. 8.6: Add Source dialog box.

The Add Load button

Displays the Add Load dialog box for adding a load to the selected segment in the wire, Fig. 8.7. A load can either represent a resistance in series with an inductance or a resistance in series with a capacitance.

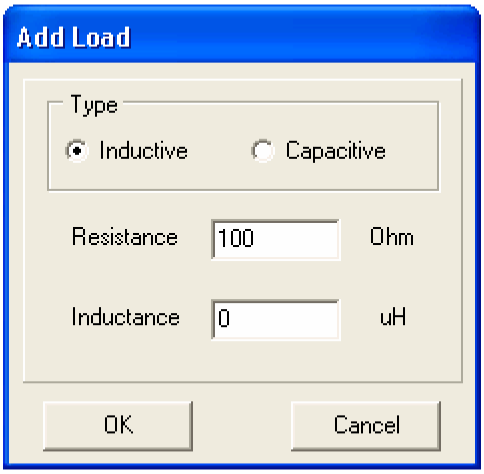


Fig. 8.7: Add Load dialog box.

The Delete button

If the selected segment has a source or a load on it, the Delete button is enabled. Pressing this button deletes the source or load placed in the segment.

The Modify button

If the selected segment has a source or a load on it, the Modify button is enabled. Pressing this button displays the Modify dialog box, where the source or load can be modified.

The Exit button

Closes the Source/Load toolbar.

8.3 Adding Sources

A discrete source or generator can be added at a given position on a selected wire by means of the following step-by-step procedure:

1. By clicking with the right mouse button in any part of a wire a pop-up menu will be shown, Fig. 8.3.
2. Choose the Source/Load command from the pop-up menu to display the Source/Load toolbar, Fig. 8.5.
3. Move the Track-bar and select a segment on the wire.
4. Press the Add Source button to display the Add Source dialog box, Fig. 8.6.
5. Choose the type of source and define its amplitude (rms value), phase and internal impedance. Then, press the OK button of the Add Source dialog box.
6. Move the Track-bar, select another segment and repeat steps 1 to 5.
7. Press the Exit button of the Source/Load toolbar.

8.4 Editing Sources

A discrete source or generator placed on a wire can be edited by means of the following step-by-step procedure:

1. By clicking with the right mouse button in any part of a wire a pop-up menu will be shown, Fig. 8.3.
2. Choose the Source/Load command from the pop-up menu to display the Source/Load toolbar, Fig. 8.5.
3. Move the Track-bar and select the segment where the source is placed.
4. Press the Modify button to display the Modify dialog box, where the source can be modified. The source can also be deleted by pressing the Delete button of the Source/Load toolbar.
5. Move the Track-bar, select another segment and repeat steps 1 to 5.
6. Press the Exit button of the Source/Load toolbar.

8.5 Adding Loads

A load impedance can be added at a given position on a selected wire by means of the following step-by-step procedure:

1. By clicking with the right mouse button in any part of a wire a pop-up menu will be shown, Fig. 8.3.
2. Choose the Source/Load command from the pop-up menu to display the Source/Load toolbar, Fig. 8.5.
3. Move the Track-bar and select a segment on the wire.
4. Press the Add Load button to display the Add Load dialog box, Fig. 8.7.
5. Choose the type of load. A load can either represent a resistance in series with an inductance or a resistance in series with a capacitance. Then, press the OK button of the Add Load dialog box.
6. Move the Track-bar, select another segment and repeat steps 1 to 5.
7. Press the Exit button of the Source/Load toolbar.

8.6 Editing Loads

A load impedance placed on a wire can be edited by means of the following step-by-step procedure:

1. By clicking with the right mouse button in any part of a wire a pop-up menu will be shown, Fig. 8.3.
2. Choose the Source/Load command from the pop-up menu to display the Source/Load toolbar, Fig. 8.5.
3. Move the Track-bar and select the segment where the load is placed.
4. Press the Modify button to display the Modify dialog box, where the load can be modified. The load impedance can also be deleted by pressing the Delete button of the Source/Load toolbar.
5. Move the Track-bar, select another segment and repeat steps 1 to 5.
6. Press the Exit button of the Source/Load toolbar.

8.7 Enabling/Disabling Loads

All of the load impedances can be enabled or disabled at the same time. This option avoids deleting load impedances placed on wire segments when loads must not be taken into account in the simulation.

Choose Simulate/Configure... in the main menu to display the Configuration dialog box. Then, select the Options page, Fig. 8.8. If the option Load Impedances in this page is checked, the loads are enabled. Uncheck the Load Impedances option in order to disable all of them.

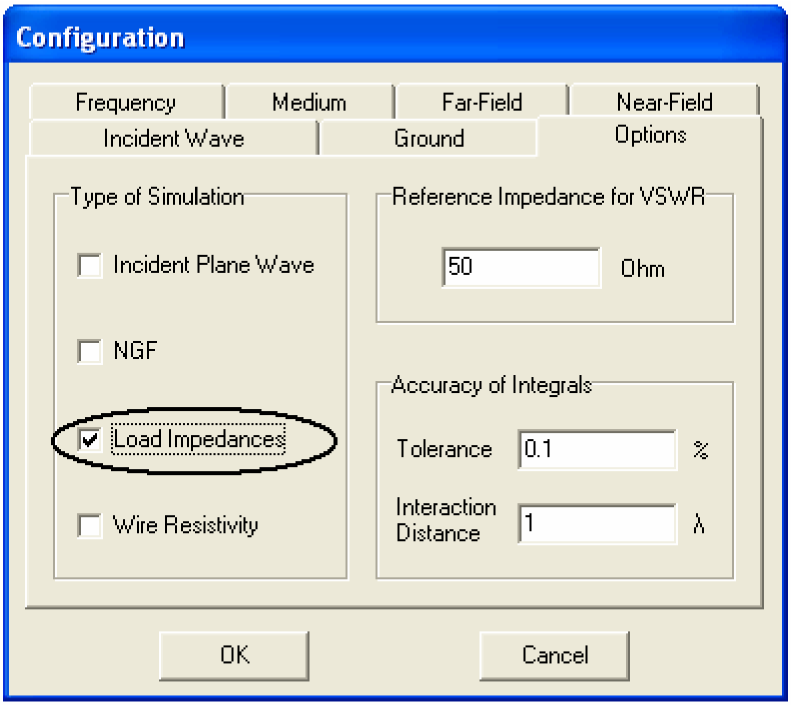


Fig. 8.8: Load impedances option in the Options page of the Configuration dialog box. If this option is checked, all of the loads are enabled, otherwise they are disabled.

9. Incident Field Excitation

9.1 Choosing an Incident Wave as Excitation

When an incident plane wave have to be used as the excitation for the wire structure, the **Incident Plane Wave** option in the Options page of the Configuration dialog box must be **checked**.

1. Choose Simulate/Configure... in the main menu to display the Configuration dialog box.
2. Select the Options page. The Incident Plane Wave option is in this page.
3. The Incident Plane Wave option must be checked, as shown in Fig. 9.1.

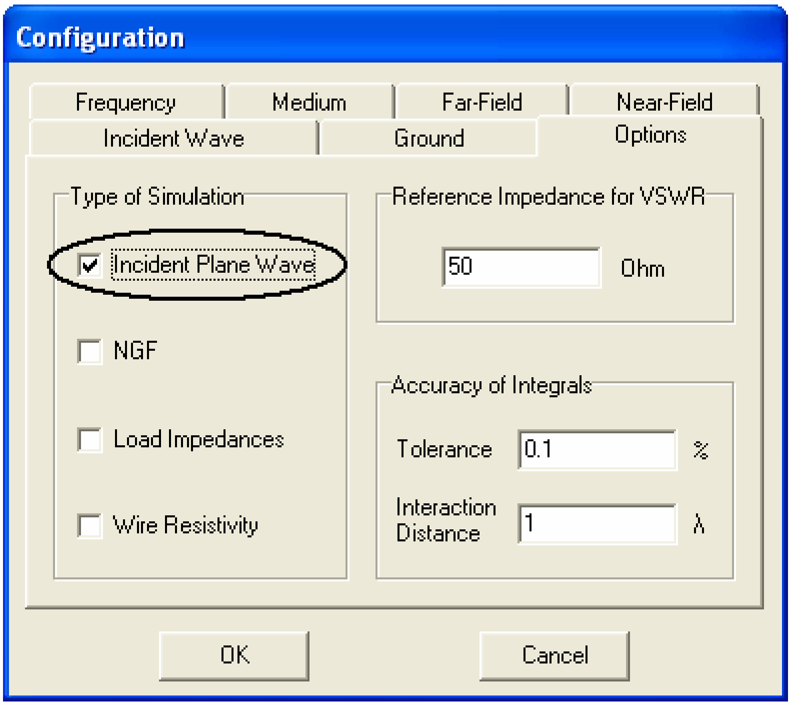


Fig. 9.1: The Incident Plane Wave option in the Options page of the Configuration dialog box must be checked when an incident plane wave have to be used as the excitation.

9.2 Defining the Incident Field

The parameters defining the incident field can be set in the Incident Wave page of the Configuration dialog box.

The following parameters have to be defined for the incident wave excitation:

- **E-Field Major Axis [V/m]:** Amplitude, in Volts per meter (rms value), of the linearly polarized incoming electric field. For elliptical polarization, it is the length of the major ellipse axis.
- **Axial Ratio**: For an elliptically polarized plane wave, it is the ratio of the minor axis to the major axis of the ellipse. A positive axial ratio defines a right-handed ellipse and a negative axial ratio defines a left-handed ellipse. If the axial ratio is set to zero, a linearly polarized plane wave is defined.
- **Phase Reference [deg]**: Phase, in degrees, of the incident plane wave at the origin of coordinates. It can be used to change the phase reference in the calculation. Its value only shifts all phases in the structure by the same amount.
- **Gamma [deg]:** Polarization angle of the incident electric field in degrees. For a linearly polarized wave, Gamma is measured from the plane of incidence to the direction of the electric field vector as it is shown in Fig. 4.11. For an elliptically polarized wave, Gamma is the angle between the plane of incidence and the major ellipse axis.
- **Theta [deg]:** Zenith angle of the incident direction in degrees.
- **Phi [deg]:** Azimuth angle of the incident direction in degrees.

See Section 4.5 for further information regarding these parameters.

Important Information

When an incident plane wave is used as excitation, all discrete sources, if any, will not be considered in the simulation.

9.3 Using the 3D-View User Interface

This user interface allows entering the parameters of the incident field in an easy way. The following step-by-step procedure describes how to use this tool:

1. Choose Simulate/Configure... in the main menu to display the Configuration dialog box.
2. Select the Incident Wave page.
3. Press the **3D View** button to open the interface and display the Incident Wave dialog box, Fig. 9.2.
4. Write the Gamma, Theta and Phi angles and press ENTER. You can also use the small arrows in the Incident Wave dialog box to change these angles.
5. Close the Incident Wave dialog box. The angles that you entered in the Incident Wave dialog box will appear in the Incident Wave page of the Configuration dialog box, Fig. 9.3.
6. Press the OK button of the Configuration dialog box.

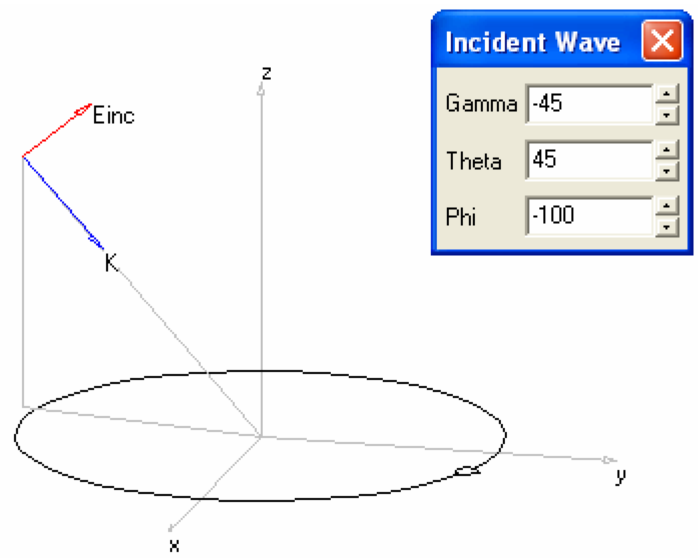


Fig. 9.2: 3D View user interface for the incident wave definition. It is also shown the Incident Wave dialog box. The Gamma, Theta and Phi angles are set to –45, 45 and –100 degrees, respectively.

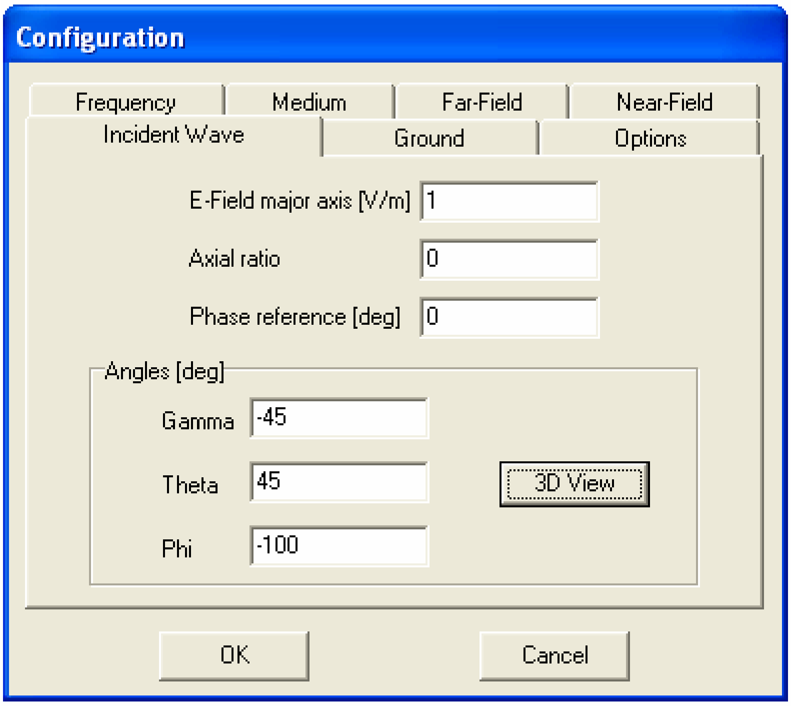


Fig. 9.3: The Gamma, Theta and Phi angles entered in the Incident Wave dialog box will appear in the Incident Wave page of the Configuration dialog box.

10. Ground Connections

10.1 Adding a PEC Ground Plane

A perfectly electric conducting (PEC) ground plane, parallel to the xy-plane, can be added to the model by means of the following procedure:

1. Choose Simulate/Configure... in the main menu to display the Configuration dialog box.
2. Select the Ground page.
3. Check the **Perfect** option, Fig. 10.1.
4. Specify the ground plane position under the **Height** label.
5. Press the OK button of the Configuration dialog box.

When the perfect ground is selected, an infinite PEC ground plane will be placed at the specified position from the xy-plane. The PEC ground position is given by the following values of Height:

- ✓ If Height is positive the PEC ground plane will be above the xy-plane.
- ✓ If Height is set to zero the PEC ground plane will be on the xy-plane.
- ✓ If Height is negative the PEC ground plane will be below the xy-plane.

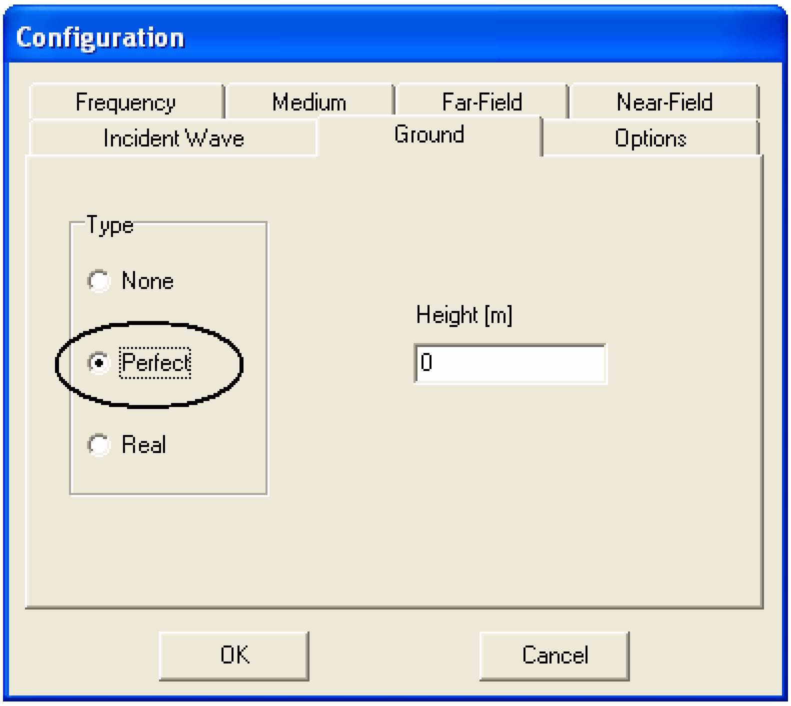


Fig. 10.1: Perfect option in the Ground page of the Configuration dialog box. The ground plane position is given by the value of Height.

10.2 Adding a Real Ground Plane

A real ground plane, located on the xy-plane, can be added to the model by means of the following procedure:

1. Choose Simulate/Configure... in the main menu to display the Configuration dialog box.
2. Select the Ground page.
3. Check the **Real** option, Fig. 10.2.
4. Specify the ground **Permittivity**, **Permeability** and **Conductivity**.
5. Press the OK button of the Configuration dialog box.

When the real ground is selected, the currents flowing on the wire structure are computed using a PEC ground plane. The real ground is only considered in the calculation of the near- and far-fields radiated from the structure. Near-fields are obtained by using the Sommerfeld-Norton approximation and the far-field is the asymptotic solution given by Fresnel's reflection coefficients. When an incident plane wave is used as the excitation of the structure, the incident wave is also affected by the reflection coefficients.

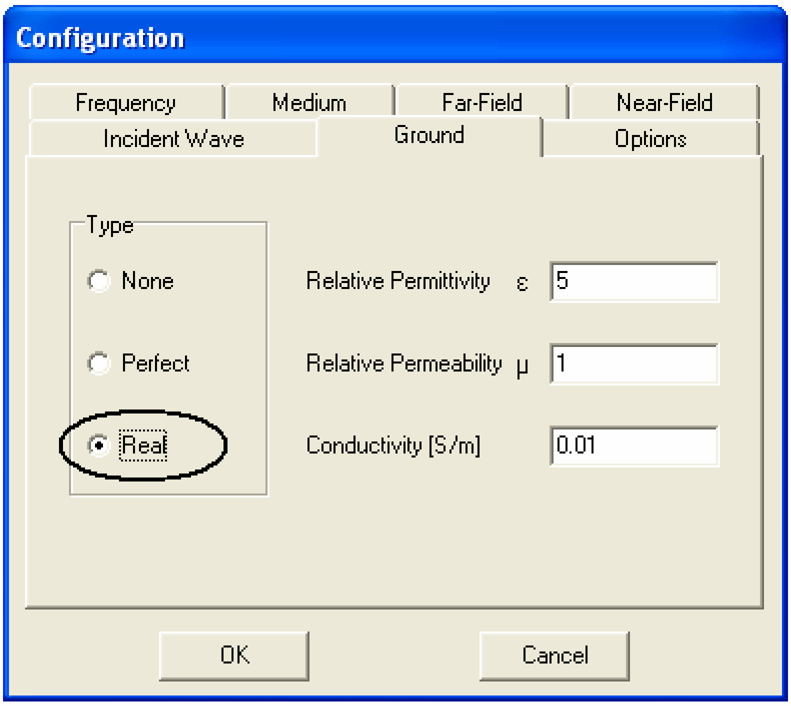


Fig. 10.2: Real option in the Ground page of the Configuration dialog box.

10.3 Connecting Wires to the Ground

Any wire has two ends: a starting point, called “Start Point”, and an ending point, called “End Point”.

A wire is automatically connected to the ground whenever the z-coordinate of a wire end is identical to the ground plane position. When a PEC ground plane is chosen, the ground position is specified by the value of Height in the Ground page of the Configuration dialog box (see Section 10.1). When a real ground is chosen, the ground position is z = 0 (xy-plane).

All wires must be placed above the ground when a ground plane is added.

The ground point positions are shown with special 3D-symbols, Fig. 10.3.

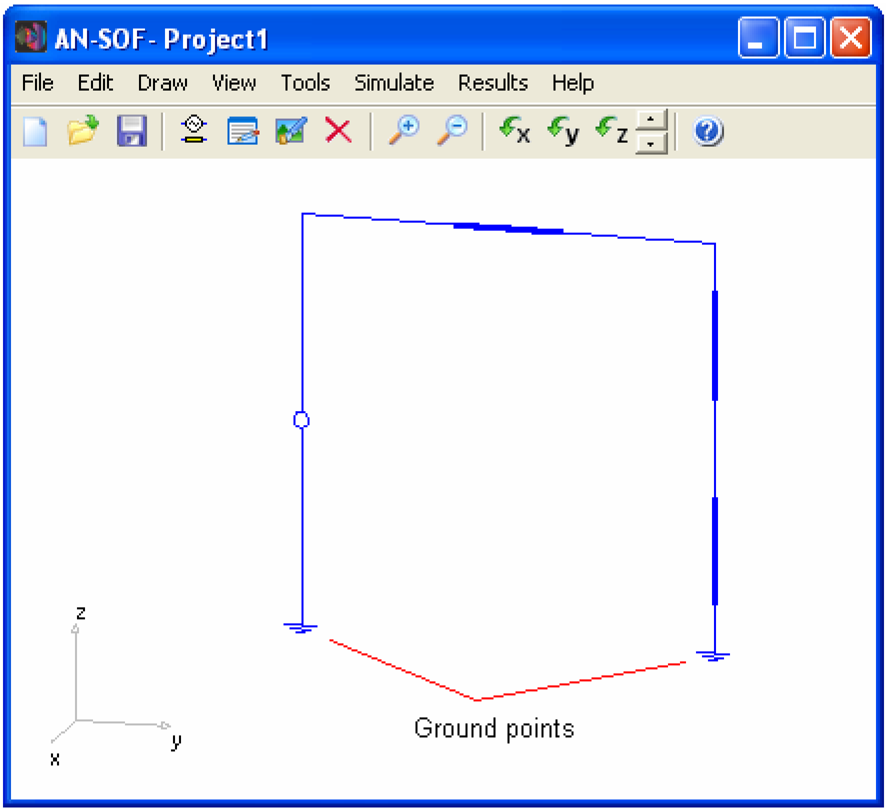


Fig. 10.3: 3D symbols used by the program for showing the ground point positions.

10.4 Removing the Ground Plane

The ground plane can be removed by means of the following procedure:

1. Choose Simulate/Configure... in the main menu to display the Configuration dialog box.
2. Select the Ground page.
3. Choose the **None** option, Fig. 10.4.
4. Press the OK button of the Configuration dialog box.

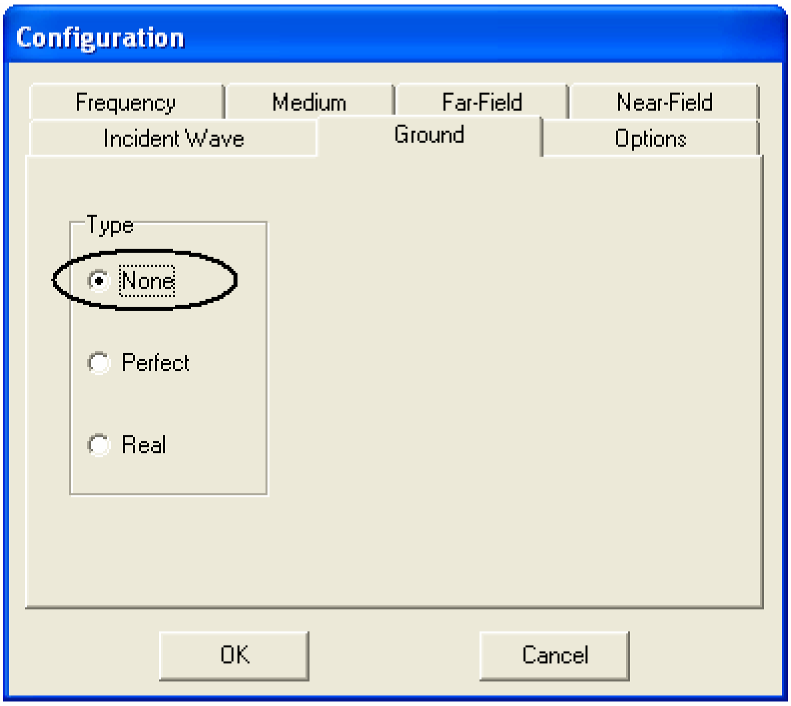


Fig. 10.4: None option in the Ground page of the Configuration dialog box.

11. 3D-Tools on the Workspace

11.1 Workspace Visualization Options

The workspace backgound can either be black or white. When a black workspace is chosen, any new wire will be white by default until a different color is specified. On the contrary, new wires are black by default when the workspace is white. The workspace color can be set pressing Edit/Preferences in the main menu and choosing the WorkSpace page tab. The color of selected wires and wire grids can be changed at any time via Edit/Wire Color in the main menu.

The width of the line used for drawing wires and axes on the workspace can be changed by selecting a Pen Width option in the WorkSpace page of the Preferences dialog box. There are three Pen Width levels: Thin, Medium and Thick. Figure 11.1 illustrates the different combinations between the workspace color and pen width.

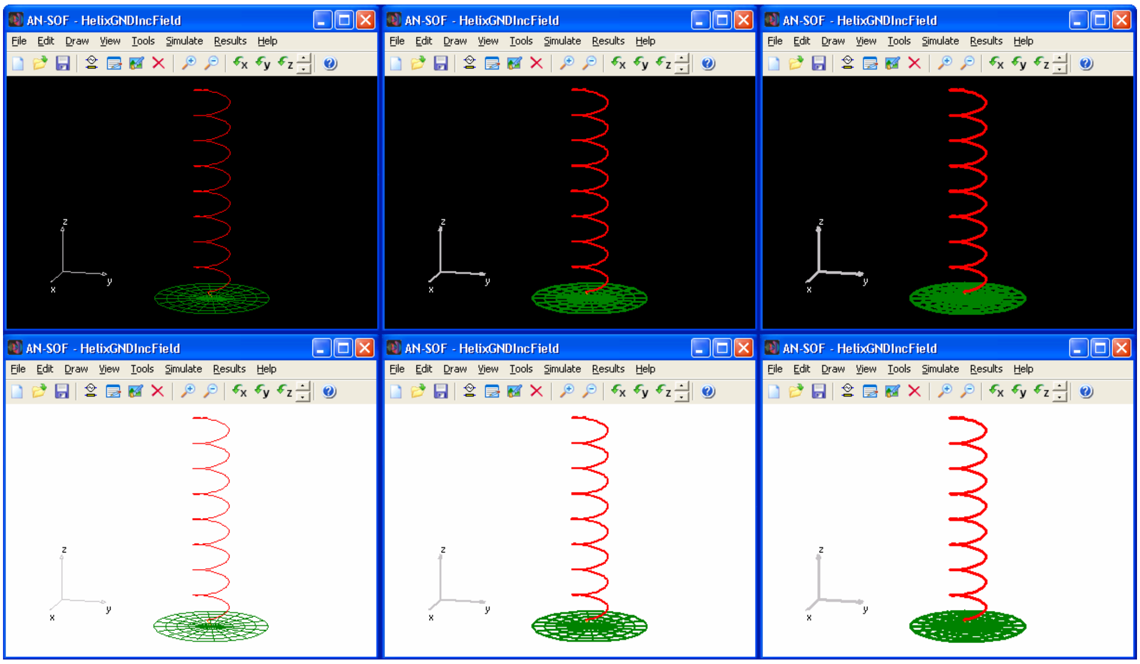


Fig. 11.1: Different visualization options in the workspace.

11.2 Viewing 3D Axes

Choose View/Axes in the main menu to display the Axes dialog box, Fig. 11.2. This dialog box allows us changing the appearance of the axes on the workspace.

There are two kinds of axes: Small and Main Axes. The Small Axes are shown on the bottom left corner of the workspace, while the Main Axes are shown at the center of the screen.

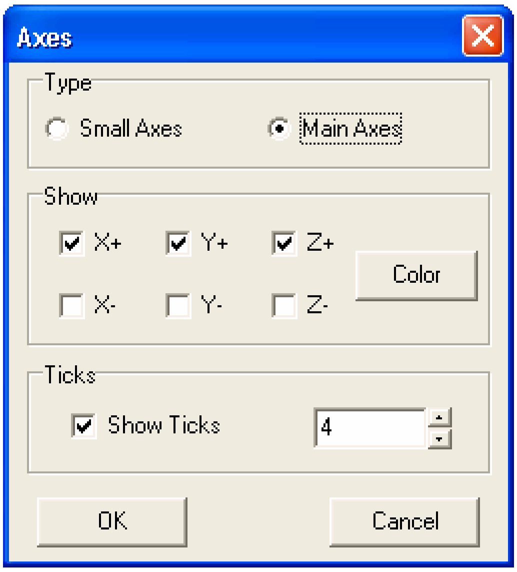


Fig. 11.2: Axes dialog box. Positive and negative axes can be displayed.

The Axes dialog box has the following components:

The Type box
Allows choosing between Small and Main Axes.

The Show box
Allows choosing the axes that will be shown on the screen.
Pressing the Color button displays a Windows® dialog box for changing the color of the Main Axes.

The Ticks box
Checking the Show Ticks option adds the specified number of ticks to the Main Axes.

Press F3 on the keyboard for switching between Small and Main Axes. The Main Axes are only available when at least one wire has been drawn on the workspace.

11.3 Zooming the Structure

Pressing the View/Zoom In command in the main menu will increase the size of the structure. This command is also available in the Edit toolbar and pressing the "+" key on the keyboard.

Pressing the View/Zoom Out command in the main menu will decrease the size of the structure. This command is also available in the Edit toolbar and pressing the "-" key on the keyboard.

11.4 Rotating the Structure

The structure can be rotated by first pressing one of the following buttons in the Edit toolbar:

Rotate X
Enables the wire structure rotation around the x-axis.

Rotate Y
Enables the wire structure rotation around the y-axis.

Rotate Z
Enables the wire structure rotation around the z-axis.

Next, press the arrows in the following button of the Edit toolbar:

Rotate
Performs a right-handed rotation of the wire structure around the selected axis when the upper arrow is pressed, and a left-handed rotation when the lower arrow is pressed.

The structure can also be rotated by pressing the following keys on the keyboard:

- ✓ **Q:** Rotates the structure around the positive x-axis.
- ✓ **A:** Rotates the structure around the negative x-axis.
- ✓ **W:** Rotates the structure around the positive y-axis.
- ✓ **S:** Rotates the structure around the negative y-axis.
- ✓ **E:** Rotates the structure around the positive z-axis.
- ✓ **D:** Rotates the structure around the negative z-axis.

11.5 Moving the Structure

Translation can be accomplished with the mouse. Moving the mouse cursor with the left mouse button pressed will move the structure on the workspace.

Double clicking the left mouse button on any part of the workspace will center the structure on it.

12. Performing the Computation

12.1 Running the Simulation

When the configuration, the geometry and the excitation are defined, AN-SOF® is ready to compute the currents flowing on the wire structure segments, as well as the radiated far- and near-fields once the currents have been obtained.

Pressing Simulate/Run ALL in the main menu can run the computation of the currents, far- and near-fields, Fig. 12.1.

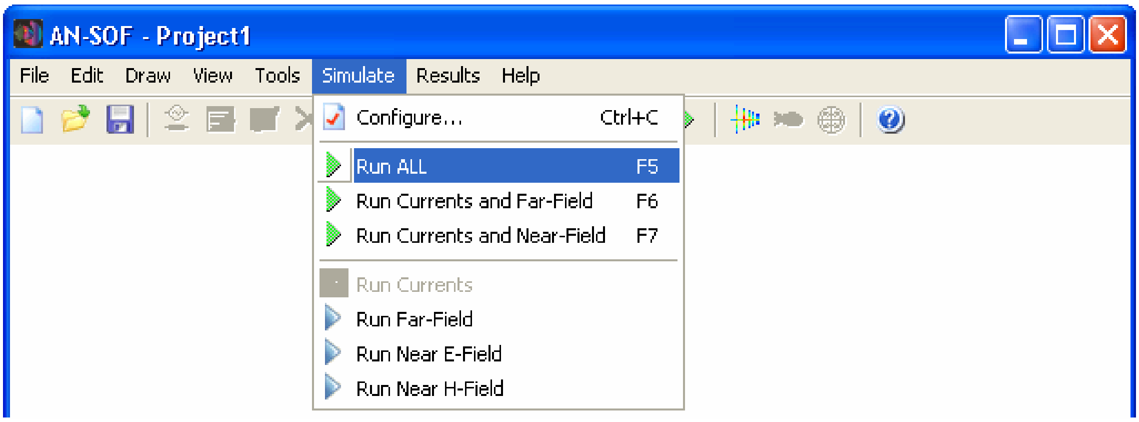


Fig. 12.1: The Simulate/Run ALL command in the main menu.

If near-fields are not required, the simulation can only be run for currents and far-fields by pressing Simulate/Run Currents and Far-Field.

If far-fields are not required, the simulation can only be run for currents and near-fields by pressing Simulate/Run Currents and Near-Field.

If needed, the currents, far- and near-fields can be computed separately as it is explained in the next sections.

12.2 Computing Currents

When the configuration, the geometry and the excitation are defined, AN-SOF® is ready to compute the currents flowing on the wire structure segments.

Pressing Simulate/Run Currents in the main menu can run the computation of the currents, Fig. 12.2.

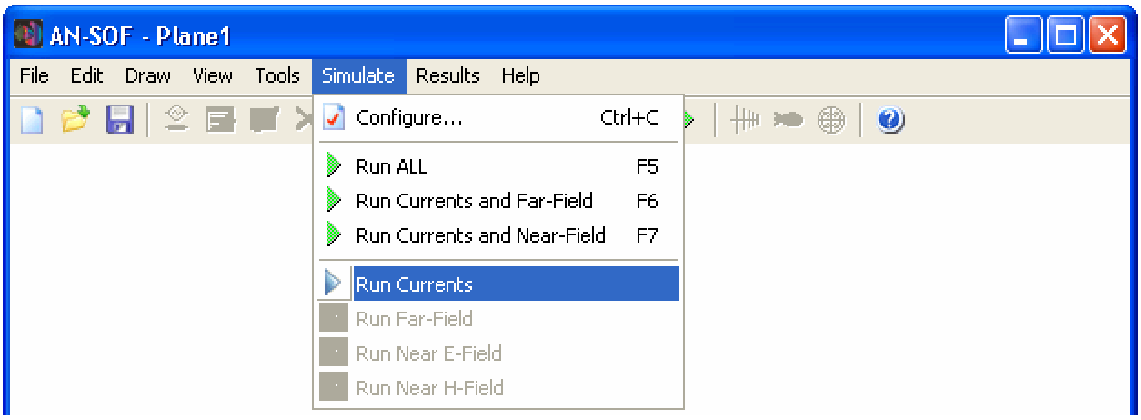


Fig. 12.2: The Simulate/Run Currents command in the main menu.

12.3 Computing Far-Fields

Once the current distribution on the wire structure has been obtained, the far-field in the angular ranges set in the Configuration dialog box can be computed.

Choosing Simulate/Run Far-Field in the main menu can run the computation of the far-field, Fig. 12.3. This option is enabled when the currents have been computed in a previous simulation.

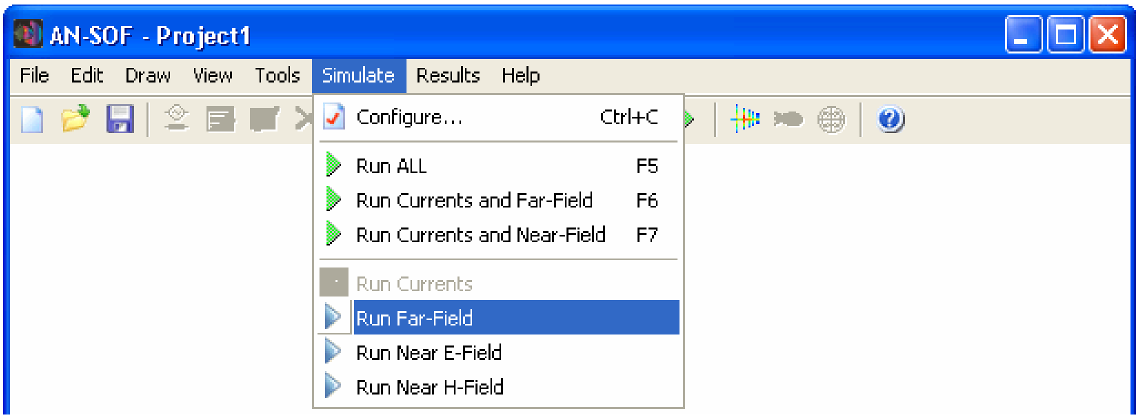


Fig. 12.3: The Simulate/Run Far-Field command in the main menu.

12.4 Computing Near Electric Fields

Once the current distribution on the wire structure has been obtained, the near electric field at points in space set in the Configuration dialog box can be computed.

Choosing Simulate/Run Near E-Field in the main menu can run the computation of the near electric fields, Fig. 12.4. This option is enabled when the currents have been computed in a previous simulation.

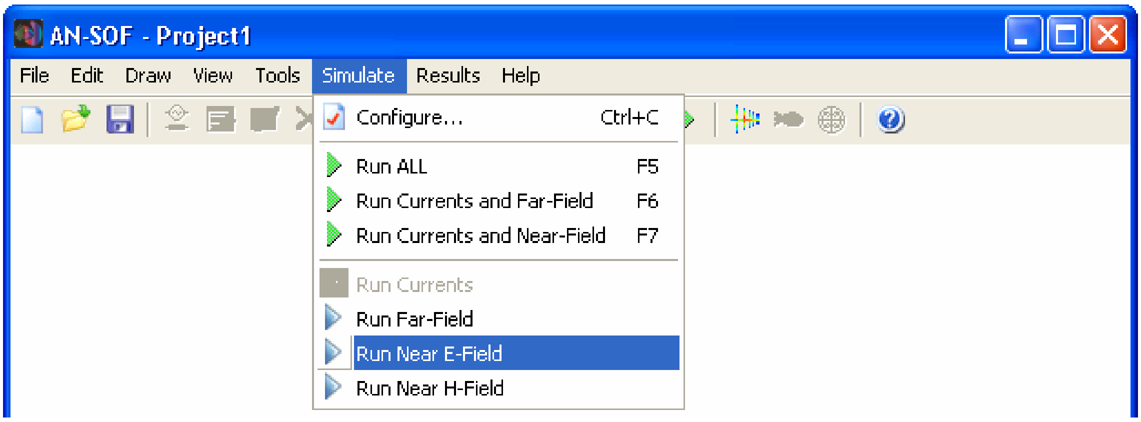


Fig. 12.4: The Simulate/Run Near E-Field command in the main menu.

12.5 Computing Near Magnetic Fields

Once the current distribution on the wire structure has been obtained, the near magnetic field at points in space set in the Configuration dialog box can be computed.

Choosing Simulate/Run Near H-Field in the main menu can run the computation of the near magnetic fields, Fig. 12.5. This option is enabled when the currents have been computed in a previous simulation.

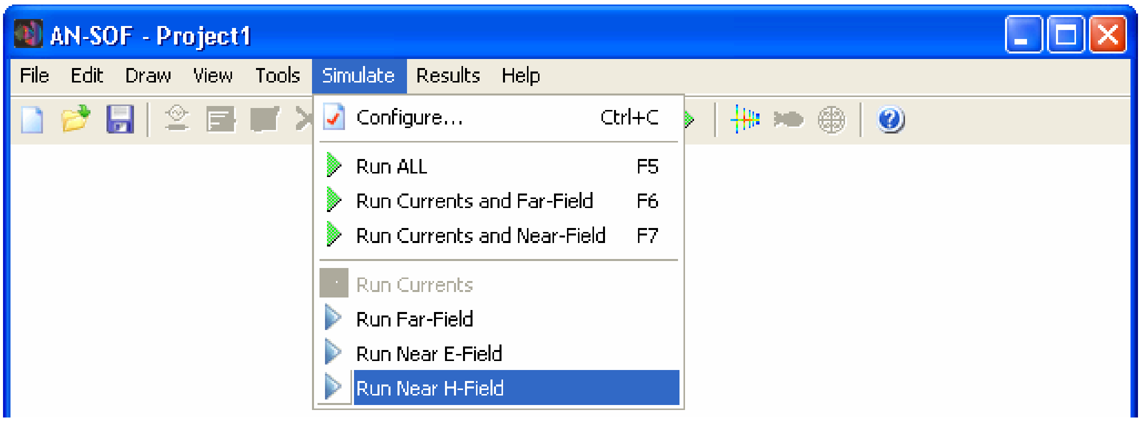


Fig. 12.5: The Simulate/Run Near H-Field command in the main menu.

12.6 Aborting the Computation

During a simulation the present frequency is shown in the Processing... dialog box, which appears when the computation is started, Fig. 12.6.

The computation can be aborted at almost any time by pressing the Abort button. The exception is when a Matlab® Component Runtime (MCR) process is running. In this case the Abort button will be disabled.

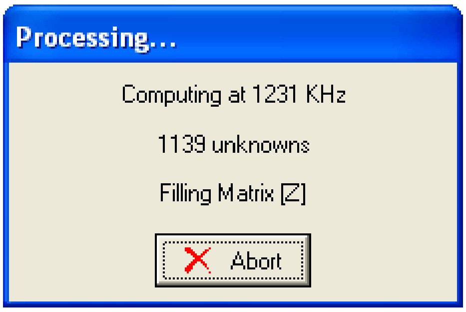


Fig. 12.6: Processing… dialog box.

12.7 Numerical Green's Function

When a Numerical Green's Function (NGF) calculation is performed, the LU-decomposed matrix of the system is stored in a file in the first computation. Then, by using the stored information, new simulations are performed faster than the first one. The NGF option can be checked in the Options page of the Configuration dialog box, as Fig. 12.7 shows.

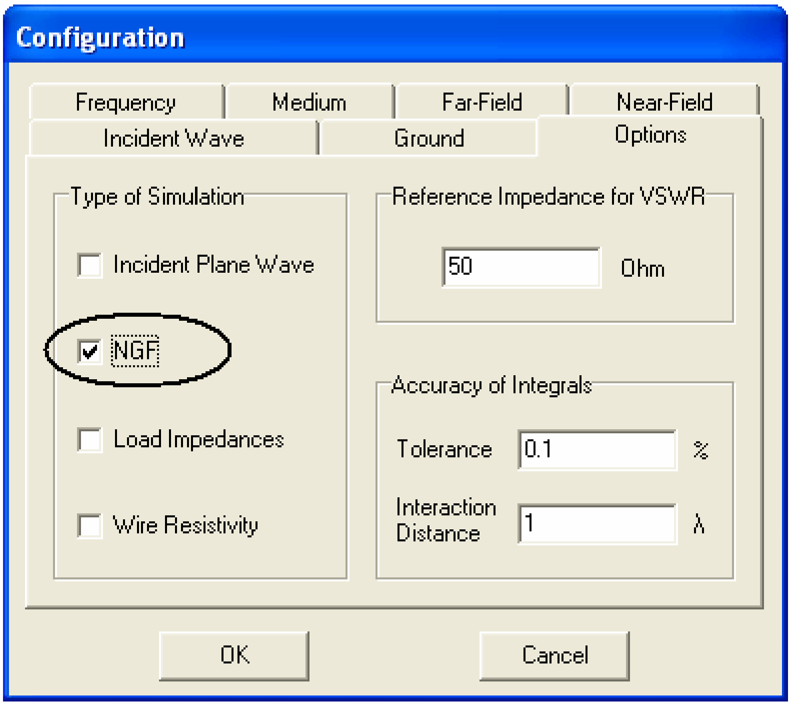


Fig. 12.7: NGF option in the Options page of the Configuration dialog box.

13. Visualizing the Computed Results

The results from a simulation can be visualized using the Results menu commands. These commands allow us performing two operations:

- ✓ ***Plotting results***
- ✓ ***Listing results***

13.1 Plotting Currents

A 3D plot of the current distribution over the whole wire structure can be shown by choosing Results/Plot Current Distribution in the main menu, Fig. 13.1. This command will execute the AN-3D Pattern® program where the current in amplitude will be seen as a color map scale. Additionally, the currents in phase, real, and imaginary parts can be plotted selecting these options in the main menu of AN-3D Pattern®, Fig. 13.2.

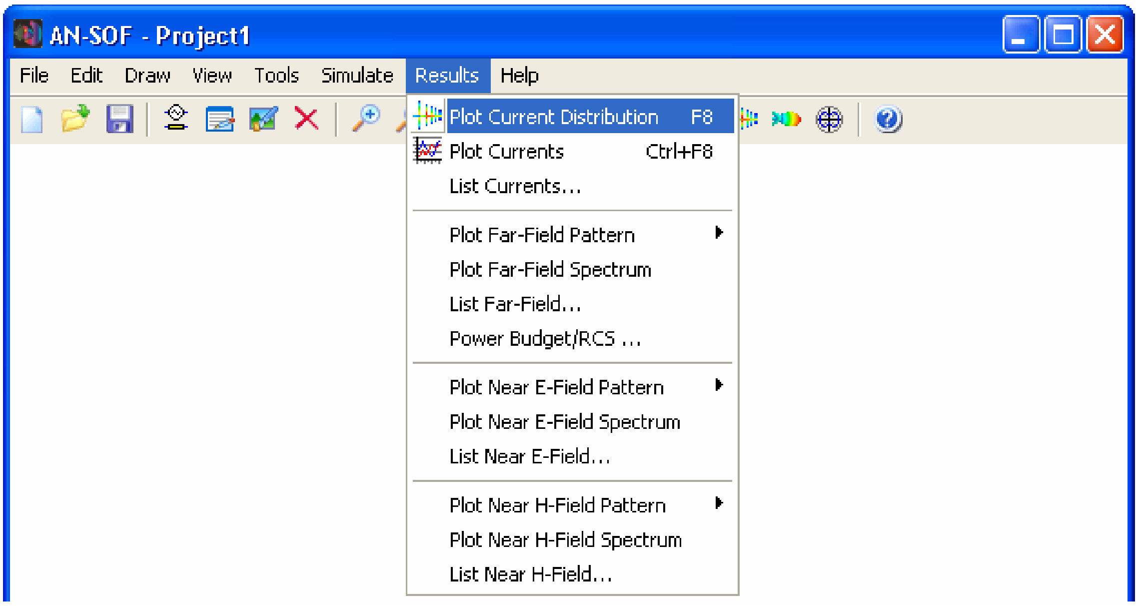


Fig. 13.1: Results/Plot Current Distribution command in the main menu.

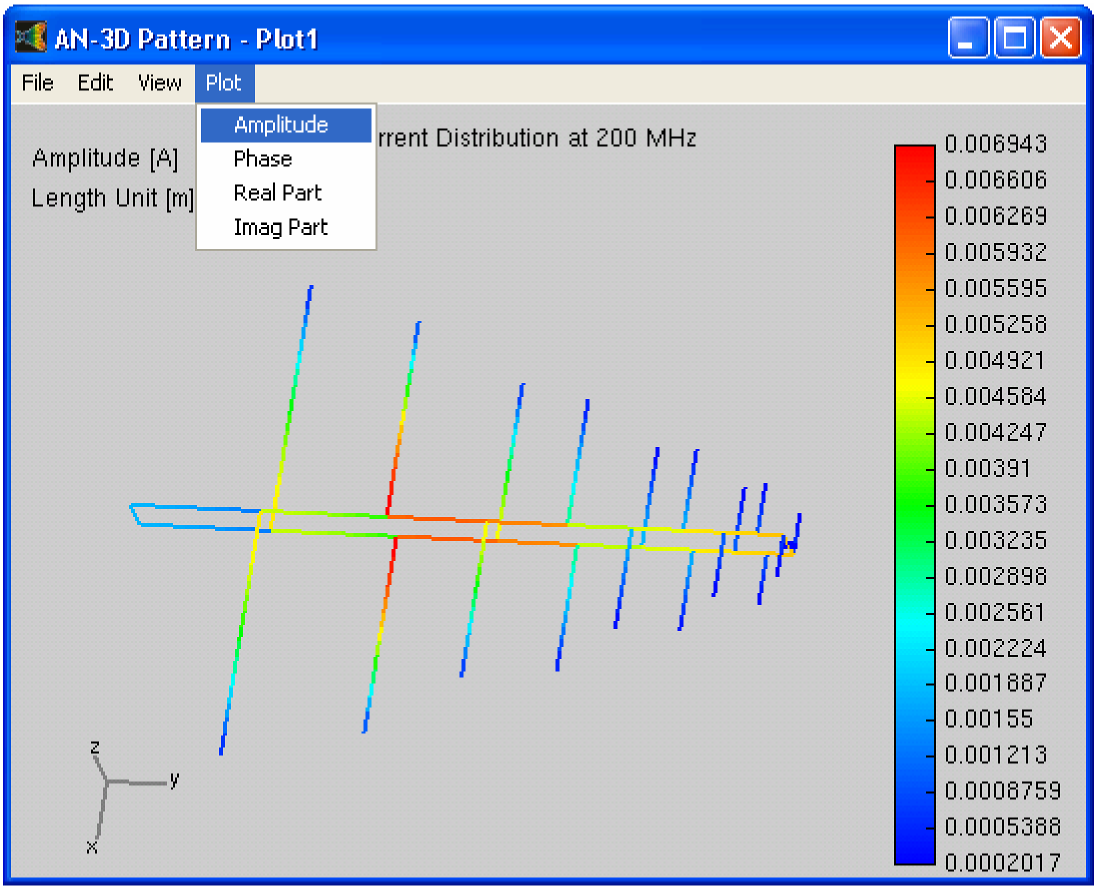


Fig. 13.2: Current distribution in amplitude plotted by AN-3D Pattern®.

A 2D plot of the current distribution along a particular wire can be shown by clicking the right mouse button on the wire, and then choosing Plot Currents from the displayed pop-up menu, Fig. 13.3.

The Plot Currents command executes the **AN-XY Chart®** program, where the current is plotted in amplitude vs. position along the selected wire, Fig. 13.4. The current distribution can also be plotted in phase, real and imaginary parts by choosing these commands under View in the AN-XY Chart® main menu.

Clicking the left mouse button on a wire and choosing Results/Plot Currents in the main menu can also plot the currents.

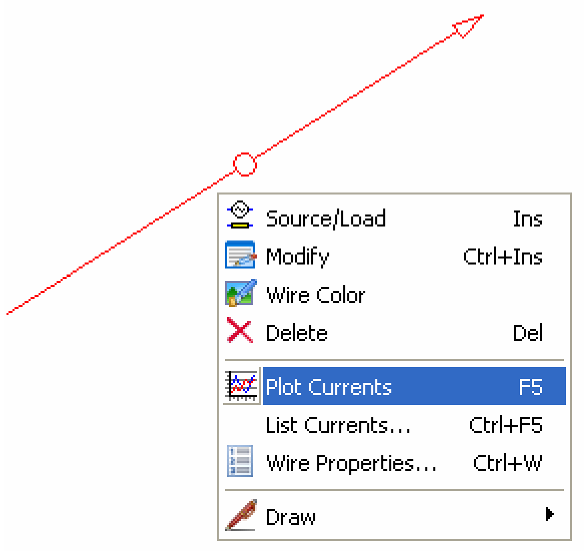


Fig. 13.3: The Plot Currents command in the pop-up menu.

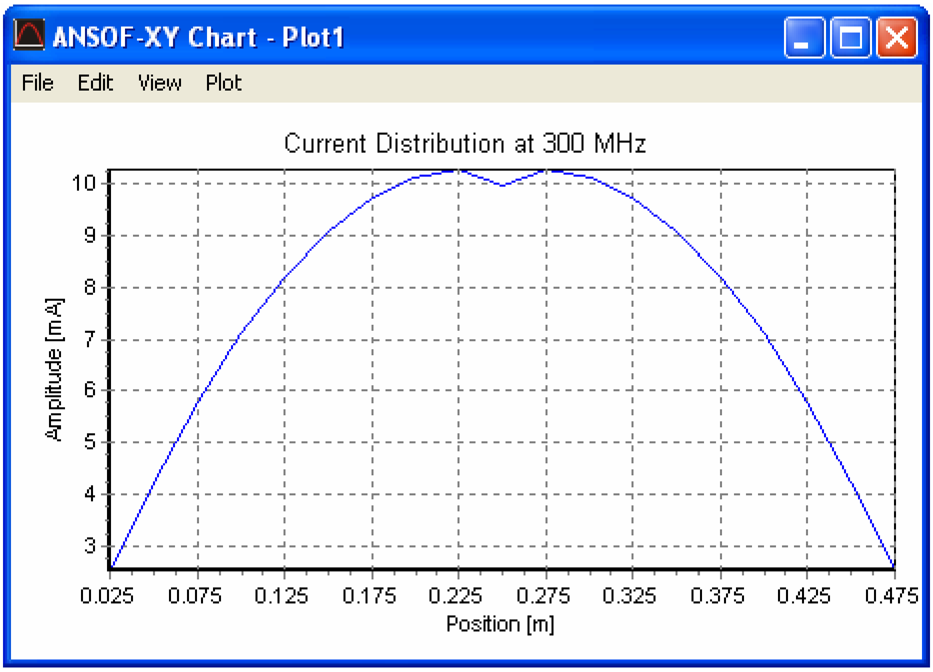


Fig. 13.4: Current distribution in amplitude plotted by AN-XY Chart®.

The graphs plotted by AN-XY Chart® can be zoomed by expanding a box with the left mouse button pressed on the plot. Also, the AN-XY Chart® main menu offers functions to change the units of the plotted magnitudes and to export data.

13.2 The List Currents Toolbar

Clicking the right mouse button on a wire shows a pop-up menu. Choosing the List Currents command from the pop-up menu shows the List Currents toolbar, Fig. 13.5. This toolbar allows selecting an individual wire segment for visualizing its currents vs. frequency. Also, if the selected segment has a source or load, the lists of input impedances, admittances, voltages, powers, reflection coefficient, VSWR, return and transmission losses are available.

The List Currents command can also be chosen by first selecting a wire by clicking the left mouse button on it, and next choosing Results/List Currents in the main menu. This command is enabled when the currents are computed.

The List Currents toolbar has the following components:

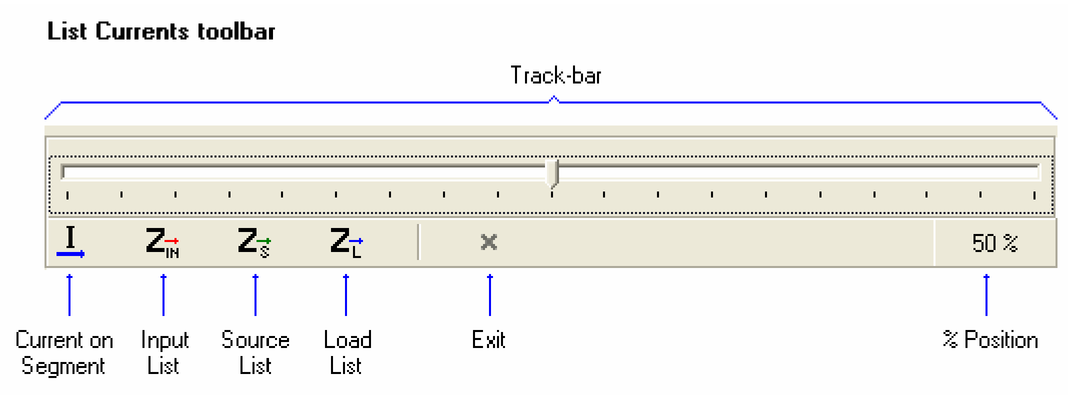


Fig. 13.5: The List Currents toolbar.

The Track-bar

Each position of the Track-bar corresponds to the position of a segment in the selected wire. Thus, the Track-bar allows us selecting a particular segment on the wire. At the right corner of the Track-bar the position of the selected segment is shown as a percentage of the wire length. The segment position is measured from the starting point of the wire to the middle point of the segment, and it is defined as follows:

$$\% \text{ position} = 100 \text{ (position / wire length)}$$

The Current on Segment button

Displays the Current on Segment dialog box, Fig. 13.6, which shows a list of the currents in the selected segment versus frequency. The frequency spectrum of the current in the selected segment can be plotted pressing the Plot button.

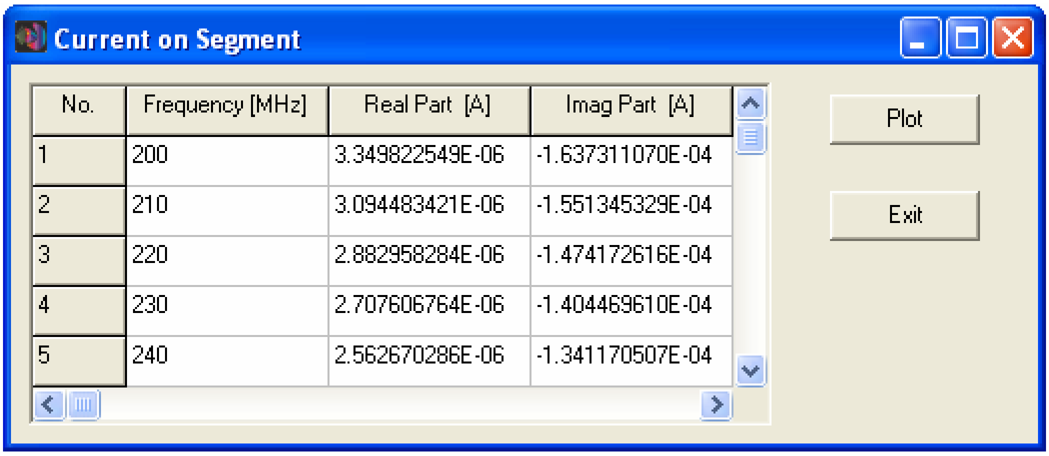


Fig. 13.6: The Current on Segment dialog box.

The Input List button

If the selected segment has a source on it, the Input List button is enabled. Pressing this button displays the Input List dialog box, Fig. 13.7. This dialog box shows the list of input impedances, admittances, currents, voltages and powers in the selected segment versus frequency. Clicking on an item in the list and pressing the Plot button will plot the chosen item as a function of frequency. The input impedance can be shown in a Smith chart by pressing the Smith button.

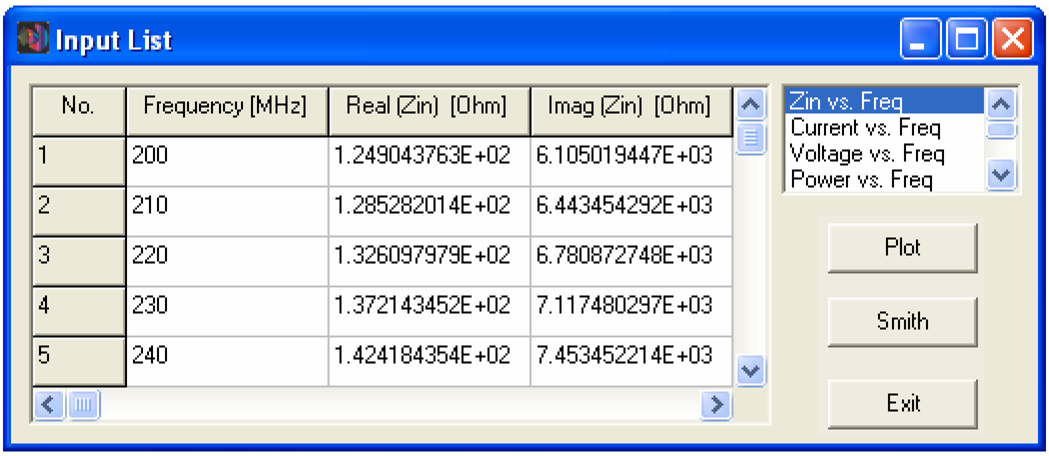


Fig. 13.7: The Input List dialog box.

The Source List button

If the selected segment has a source on it, the Source List button is enabled. Pressing this button displays the Source List dialog box, Fig. 13.8, which shows the list of currents, voltages and powers in the source internal impedance versus frequency. Clicking on an item in the list and pressing the Plot button will plot the selected item as a function of frequency.

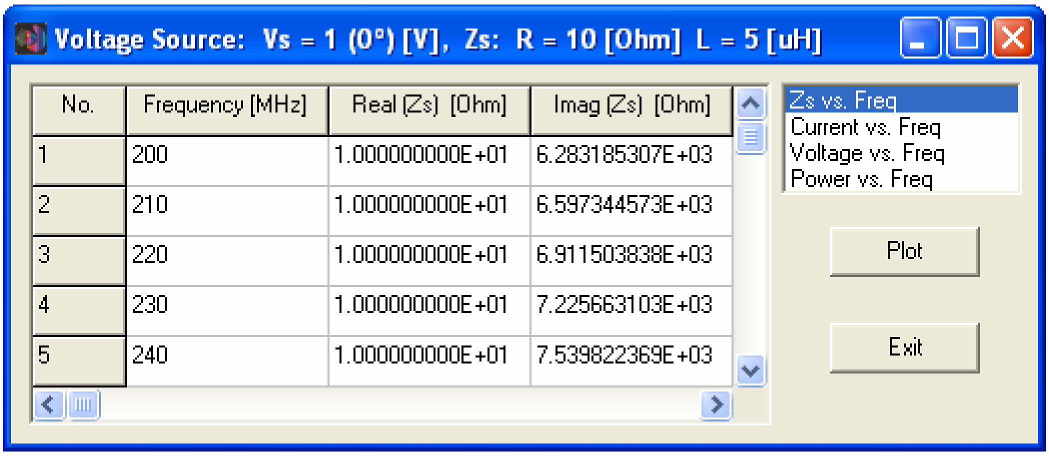


Fig. 13.8: The Source List dialog box.

The Load List button

If the selected segment has a load on it, the Load List button is enabled. Pressing this button displays the Load List dialog box, Fig. 13.9. This dialog box shows the list of load impedances, currents, voltages and powers in the selected segment versus frequency. Clicking on an item in the list and pressing the Plot button will plot the selected item as a function of frequency.

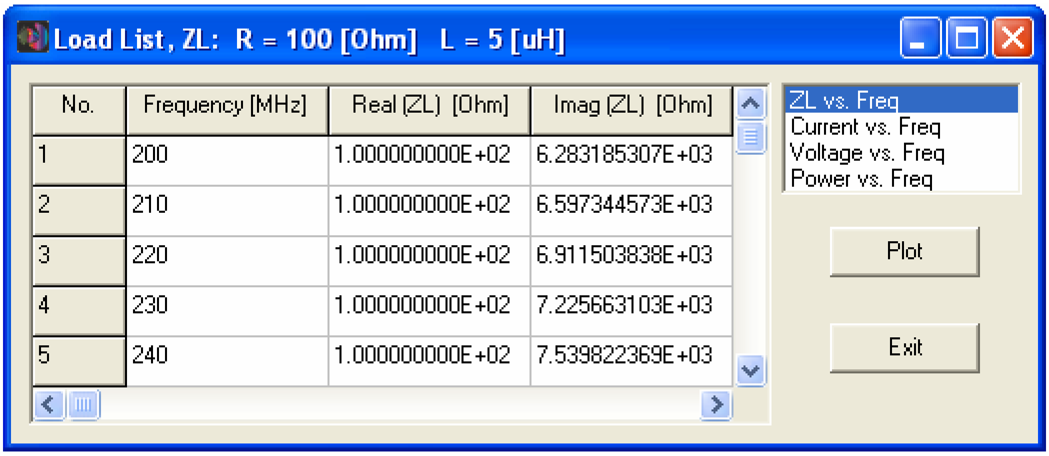


Fig. 13.9: The Load List dialog box.

The Exit button

Closes the List Currents toolbar.

13.3 Listing Currents

The following step-by-step procedure allows us selecting an individual wire segment for visualizing its currents versus frequency:

1. By clicking with the right mouse button in any part of a wire a pop-up menu will be shown.
2. Choose the List Currents command from the pop-up menu to display the List Currents toolbar, Fig. 13.5.
3. Move the Track-bar and select a segment on the wire.
4. Press the Current on Segment button to display the Current on Segment dialog box, Fig. 13.6. This dialog box shows a list of the currents in the selected segment versus frequency. Currents are shown in amplitude, phase, real and imaginary parts. Pressing the Plot button will plot the current in the selected segment as a function of frequency.
5. Press the Exit button to close the Current on Segment dialog box.
6. Move the Track-bar, select another segment and repeat steps 1 to 5.
7. Press the Exit button of the List Currents toolbar.

13.4 Listing Input Impedances

The following step-by-step procedure allows us selecting an individual wire segment, with a source placed on it, for visualizing the input impedance versus frequency:

1. By clicking with the right mouse button in any part of a wire a pop-up menu will be shown.
2. Choose the List Currents command from the pop-up menu to display the List Currents toolbar, Fig. 13.5.
3. Move the Track-bar and select a segment on the wire. The selected segment must have a source placed on it.
4. Press the Input List button to display the Input List dialog box, Fig. 13.7. This dialog box shows the list of input impedances, admittances, currents, voltages, powers, reflection coefficient, VSWR, return and transmission losses in the selected segment versus frequency. Selecting an item from the list and pressing the Plot button will plot the selected item as a function of frequency. The reference impedance for reflection and VSWR calculations is defined in the Options page of the Configuration dialog box. Besides, the imput impedance can be plotted in a Smith chart by pressing the Smith button.
5. Press the Exit button to close the Input List dialog box.
6. Move the Track-bar, select another segment and repeat steps 1 to 5.
7. Press the Exit button of the List Currents toolbar.

13.5 Showing Smith Charts

The input impedance as a function of frequency can be plotted in a Smith chart , which is accessible from the Input List dialog box, Fig. 13.7. By pressing the Smith button provided in this dialog box, the AN-Smith® program will be executed to plot the input impedance in a Smith chart. Therefore, follow the procedure described in the previous section for listing the input impedances versus frequency, and then press the Smith button in the List dialog box.

By clicking with the left mouse button on the impedance curve in the Smith chart, the frequency, input impedance (Zin), reflection coefficient (Rho) and VSWR will appear in a hint message, Fig 13.10. The input admittance can be plotted by selecting Plot/Admittance in the AN-Smith® main menu.

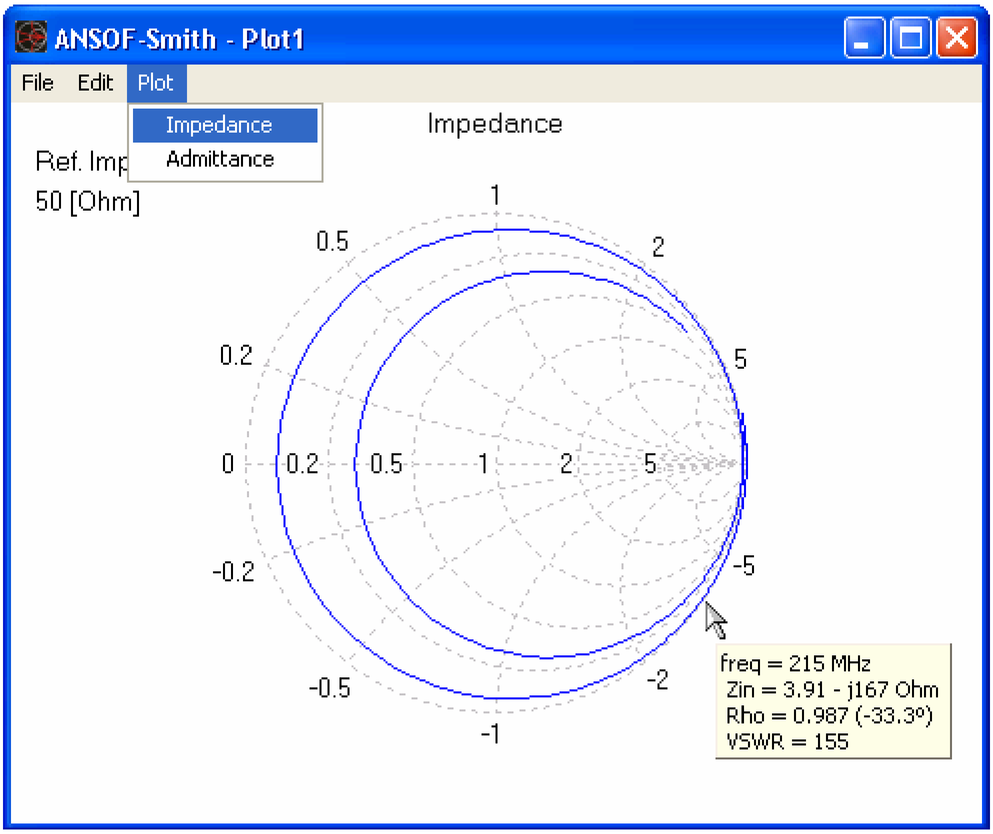


Fig. 13.10: Input impedance curve in the Smith chart plotted by the AN-Smith® program.

13.6 Listing Generator Impedances

The following step-by-step procedure allows us selecting an individual wire segment, with a source or generator placed on it, for visualizing its internal impedance versus frequency:

1. By clicking with the right mouse button in any part of a wire a pop-up menu will be shown.
2. Choose the List Currents command from the pop-up menu to display the List Currents toolbar, Fig. 13.5.
3. Move the Track-bar and select a segment on the wire. The selected segment must have a source placed on it.
4. Press the Source List button to display the Source List dialog box, Fig. 13.8. This dialog box shows the list of currents, voltages and powers in the internal impedance of the source versus frequency. Selecting an item from the list and pressing the Plot button will plot the selected item as a function of frequency.
5. Press the Exit button to close the Source List dialog box.
6. Move the Track-bar, select another segment and repeat steps 1 to 5.
7. Press the Exit button of the List Currents toolbar.

13.7 Listing Load Impedances

The following step-by-step procedure allows us selecting an individual wire segment, with a load impedance placed on it, for visualizing its behavior versus frequency:

1. By clicking with the right mouse button in any part of a wire a pop-up menu will be shown.
2. Choose the List Currents command from the pop-up menu to display the List Currents toolbar, Fig. 13.5.
3. Move the Track-bar and select a segment on the wire. The selected segment must have a load impedance placed on it.
4. Press the Load List button to display the Load List dialog box, Fig. 13.9. This dialog box shows the list of load impedances, currents, voltages and powers in the selected segment versus frequency. Selecting an item from the list and pressing the Plot button will plot the selected item as a function of frequency.
5. Press the Exit button to close the Load List dialog box.
6. Move the Track-bar, select another segment and repeat steps 1 to 5.
7. Press the Exit button of the List Currents toolbar.

13.8 Plotting 2D Far-Field Patterns

The computed radiation pattern can be shown as a 2D rectangular plot by choosing the Results/Plot Far-Field Pattern/2D Rectangular Plot command in the main menu, Fig. 13.11.

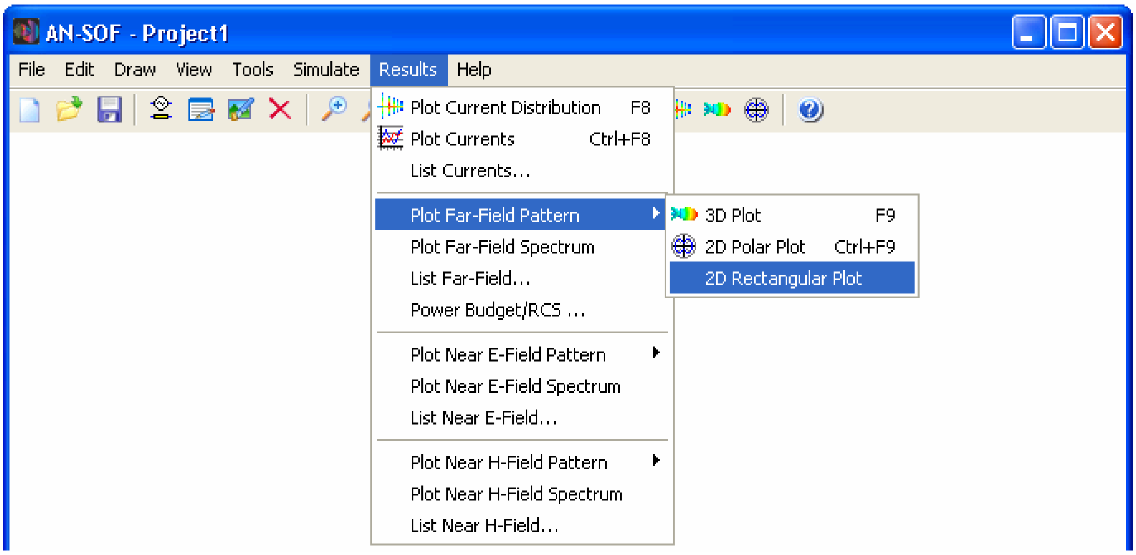


Fig. 13.11: The Results/Plot Far-Field Pattern/2D Rectangular Plot command in the main menu.

This command displays the Radiation Pattern Cut dialog box, Fig. 13.12, where two kinds of plots can be produced:

- **Conical:** Conical plots are for fixed Theta with Phi varying.
- **Vertical:** Vertical plots are for fixed Phi with Theta varying.

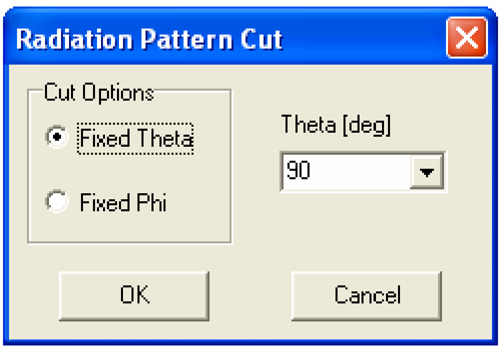


Fig. 13.12: The Radiation Pattern Cut dialog box.

Choosing a radiation pattern cut executes the AN-XY Chart® program, Fig. 13.13, where the power density or Poynting vector is plotted vs. Phi if a conical plot was chosen (for fixed Theta) or vs. Theta if a vertical plot was chosen (for fixed Phi). Selecting these options under Plot in the AN-XY Chart® main menu can also show the total E-field, the E-theta (vertical) and E-phi (horizontal) linearly polarized field components, the E-right and E-left circularly polarized components, the directivity, gain and radiation patterns. In the case of plane wave excitation, the Radar Cross Section (RCS) is plotted.

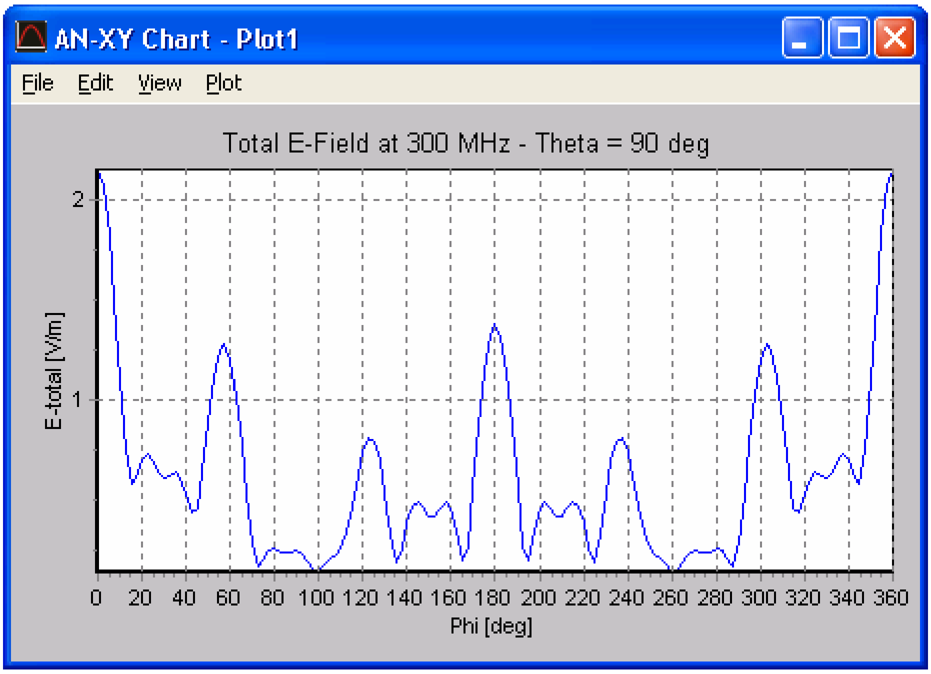


Fig. 13.13: A Radiation Pattern Cut plotted by the AN-XY Chart® program in a rectangular chart.

The far-field patterns can also be plotted in a 2D polar chart by choosing the Results/Plot Far-Field Pattern/2D Polar Plot command in the AN-SOF® main menu, Fig. 13.14. In this case, the maximun radiation, beamwith, and front to back ratio will be shown.

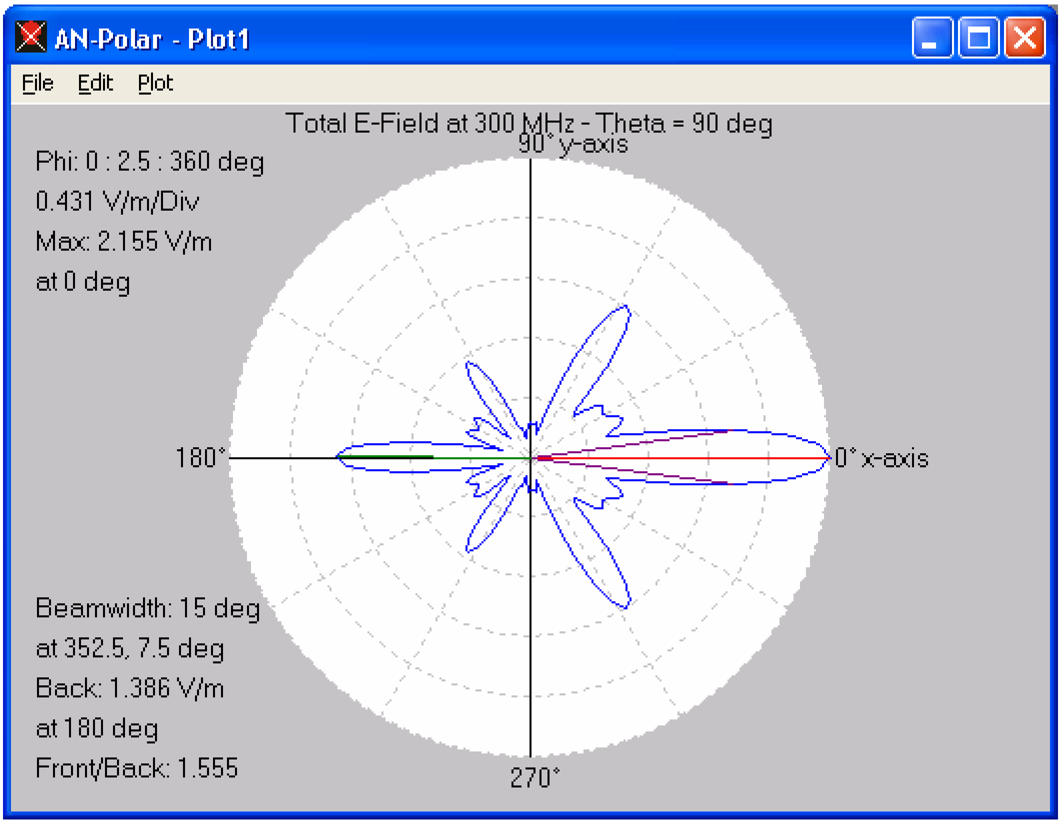


Fig. 13.14: A Radiation Pattern Cut plotted by the AN-Polar® program in a polar chart.

13.9 Plotting 3D Far-Field Patterns

The computed far-field can be shown as a 3D plot by choosing the Results/Plot Far-Field Pattern/3D Plot command in the main menu. This command executes the AN-3D Pattern® program, where the power density or Poynting vector is plotted in a 3D diagram.
The directivity, gain, radiation pattern, total E-field, E-theta (vertical) and E-phi (horizontal) linearly polarized field components as well as the E-right and E-left circularly polarized field components can also be plotted by choosing these commands under Plot in the AN-3D Pattern® main menu, Fig 13.15. In the case of plane wave excitation, the Radar Cross Section will be plotted.

The graph plotted by AN-3D Pattern® can be zoomed, rotated and moved by pressing the following keys on the keyboard:

Key	Action
Home	Return the plot to the initial view
Left Arrow ←	Move the plot to the left
Right Arrow →	Move the plot to the right
Up Arrow ↑	Move the plot upwards
Down Arrow ↓	Move the plot downwards
+	Zoom in
-	Zoom out
Q	Rotate around +X axis
A	Rotate around -X axis
W	Rotate around +Y axis
S	Rotate around -Y axis
E	Rotate around +Z axis
D	Rotate around -Z axis
F3	Switch between Main and Small axes
F4	Switch between surface and mesh

The AN-3D Pattern® main menu includes options for changing the units of the plotted magnitudes, showing a color bar and exporting data.

Note: If discrete sources were used as the excitation of the structure, the plotted far-field is the total field, but if an incident plane wave was used as the excitation, the plotted far-field is the scattered field.

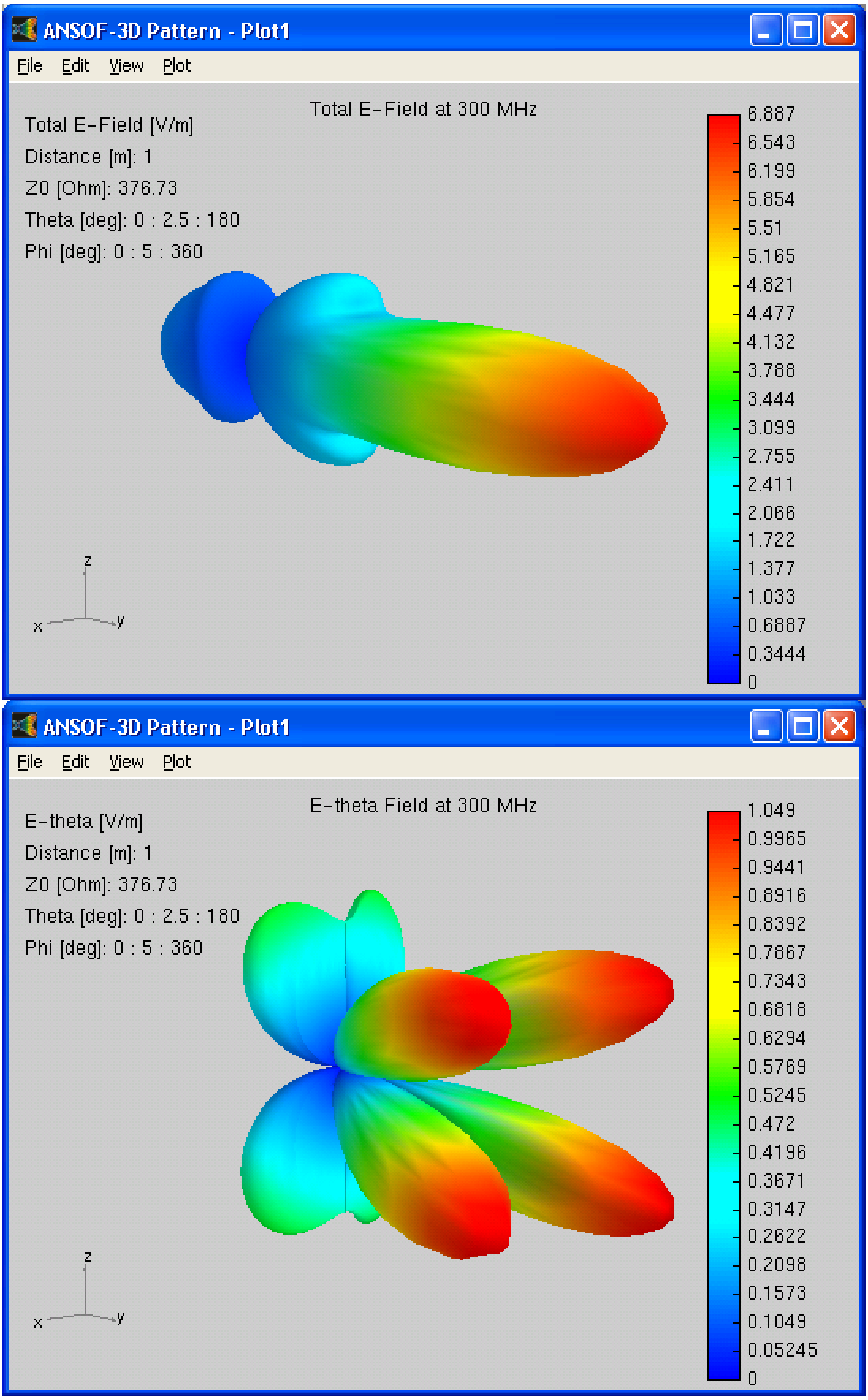


Fig. 13.15: 3D far-field patterns plotted by the AN-3D Pattern® program.

Press View/Options in the AN-3D Pattern® main menu for displaying the View Options dialog box, Fig. 13.16. Different visualization options can be chosen for the colored surface and mesh representing the radiation lobes, Fig. 13.17. Additionally, the wire structure can be shown superimposed to the radiation pattern by selecting the Wires option in the Show box.

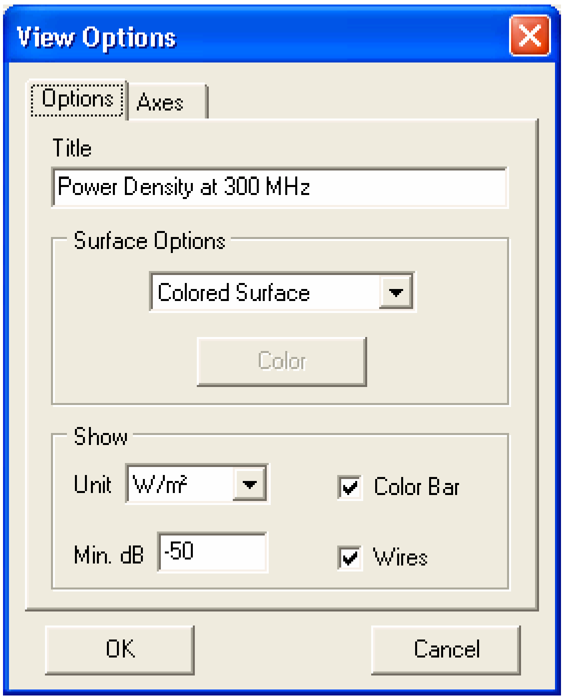


Fig. 13.16: View Options dialog box of the AN-3D Pattern® program.

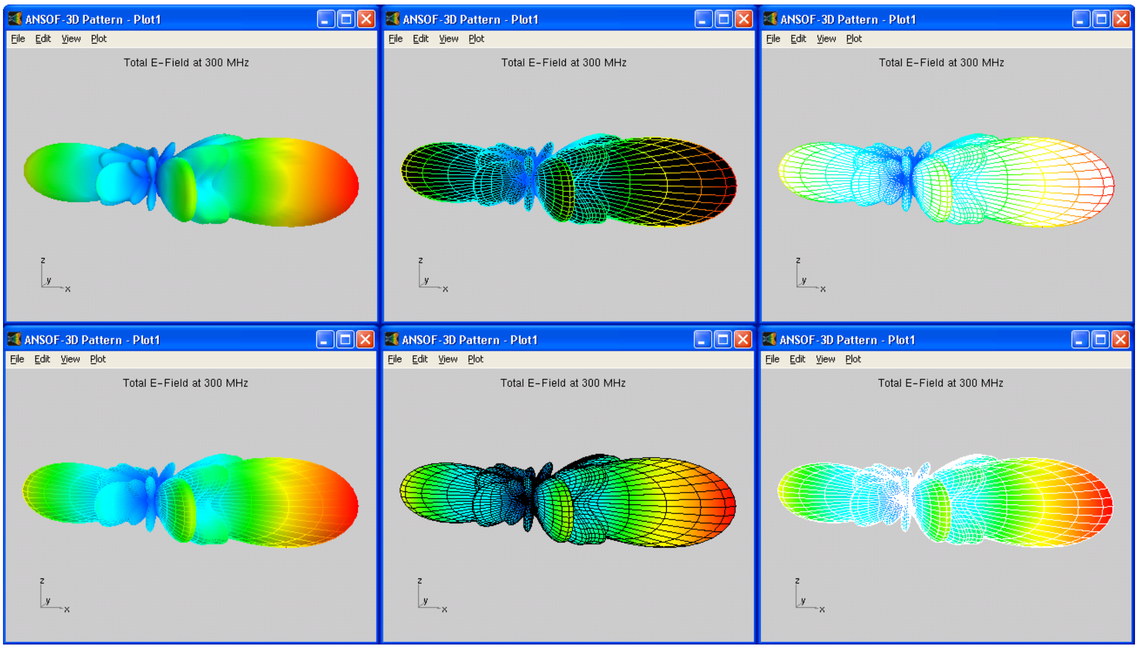


Fig. 13.17: Different visualization options for plotting radiation lobes.

13.10 Plotting Far-Field Spectra

Far-field frequency spectra are obtained when a simulation is performed by specifying a list of frequencies or a frequency sweep. For each frequency, the far-field is calculated at the several directions given by the zenith (Theta) and azimuth (Phi) angular ranges and at the distance specified in the Far-Field page of the Configuration dialog box. Therefore, in order to plot the computed far-field versus frequency, a fixed direction (Theta, Phi) must be chosen.

The far-field spectrum can be plotted via Results/Plot Far-Field Spectrum in the main menu. This command displays the Select Far-Field Point dialog box, Fig. 13.18, where the fixed Theta and Phi angles can be selected. After pressing the OK button, the AN-XY Chart® program will show the frequency spectrum of the total E-field, Fig. 13.19. The linearly polarized field components, E-theta and E-phi, as well as the circularly polarized components, E-right and E-left, can be plotted in amplitude, phase, real and imaginary parts by choosing these options under Plot in the AN-XY Chart® main menu.

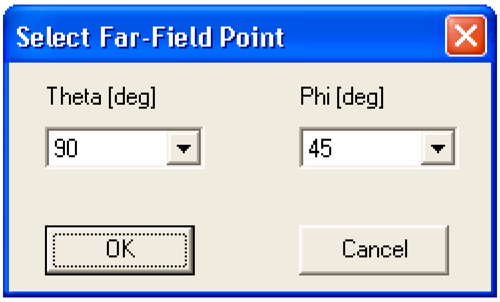


Fig. 13.18: Select Far-Field Point dialog box for selecting a fixed direction (Theta, Phi). The fixed distance is set in the Far-Field page of the Configuration dialog box.

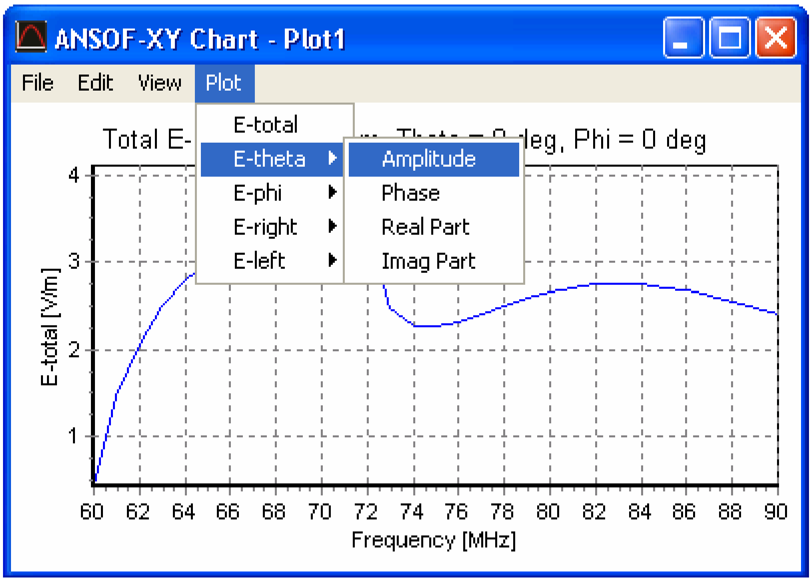


Fig. 13.19: Far-field frequency spectrum plotted by the AN-XY Chart® program.

The far-field spectrum for a given far-field point can also be listed in a table showing the computed quantities. This option is accessed via Results/List Far-Field… in the AN-SOF® main menu. The List Far-Field… command displays the Select Far-Field Point dialog box for selecting the fixed Phi and Theta angles. Then, the list of the far-field components versus frequency will be shown, which can be plotted by pressing the Plot button, Fig. 13.20.

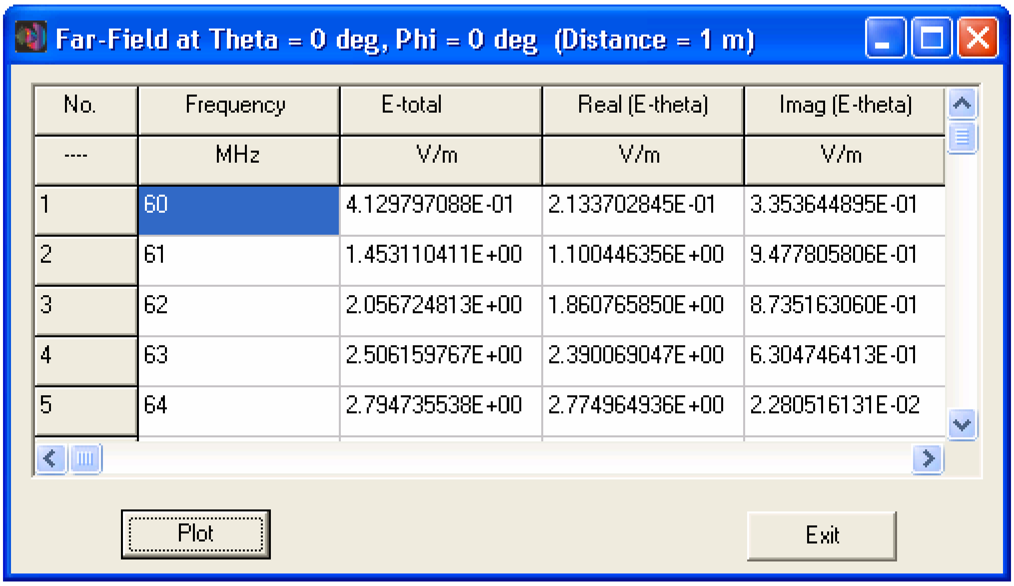


Fig 13.20: Far-Field List showing the computed quantities for all of the far-field components.

13.11 Power Budget

Choose Results/Power Budget/RCS… in the main menu to display the Power Budget dialog box, Fig. 13.21. This dialog box shows a list of the following items versus frequency when discrete generators were used as the excitation of the structure:

- The column **Input Power** shows the total input power provided by the discrete generators in the structure.
- The column **Radiated Power** shows the total radiated power from the structure.
- The column **Structure Loss** shows the total consumed power or ohmic losses in the structure.
- The column **Efficiency** is the radiated power to the input power ratio. When the structure is lossless, an efficiency of 100% is obtained.
- The column **Directivity** is the maximum value of the directivity or peak directivity of the radiating structure.
- The column **Directivity [dBi]** is the peak directivity in decibels, with an isotropic source taken as the reference.
- The column **Gain** is the maximum value of the gain or peak gain of the radiating structure.
- The column **Gain [dBi]** is the peak gain in decibels, with an isotropic source taken as the reference.
- The column **Pav** is the average power density. This value is computed averaging the power density over all directions in space.
- The column **Pmax** is the maximum value of the radiated power density.
- The columns **Theta (max)** and **Phi (max)** are the zenith and azimuth angles, respectively, in the direction of maximum radiation.
- The column **Error** is the error in the power balance of the system. One necessary, but not sufficient condition, for a valid model is that the input power to the structure be equal to the sum of the power lost and the power radiated from the structure. An error of about 1% or less in the power budget is permissible from the engineering point of view. When a real ground plane is used, this column shows the percentage of power lost in the ground due to its finite conductivity.

Select an item in the list and press the Plot button for plotting the selected item as a function of frequency.

Important Information

The average power density and the error in the power budget are meaningful quantities only if the Theta and Phi angles in the Far-Field page of the Configuration dialog box are varying in the following ranges:

If the environment is **free space** (there is no ground plane):

$0 \leq \text{Theta} \leq 180$ [degrees]

and

$0 \leq \text{Phi} \leq 360$ [degrees]

If the environment has a **ground plane**:

$0 \leq \text{Theta} \leq 90$ [degrees]

and

$0 \leq \text{Phi} \leq 360$ [degrees]

This is because the average power density must be computed averaging the power density or Poynting vector by taking into account all directions in free space. If there is a ground plane, directions must be considered in half-space.

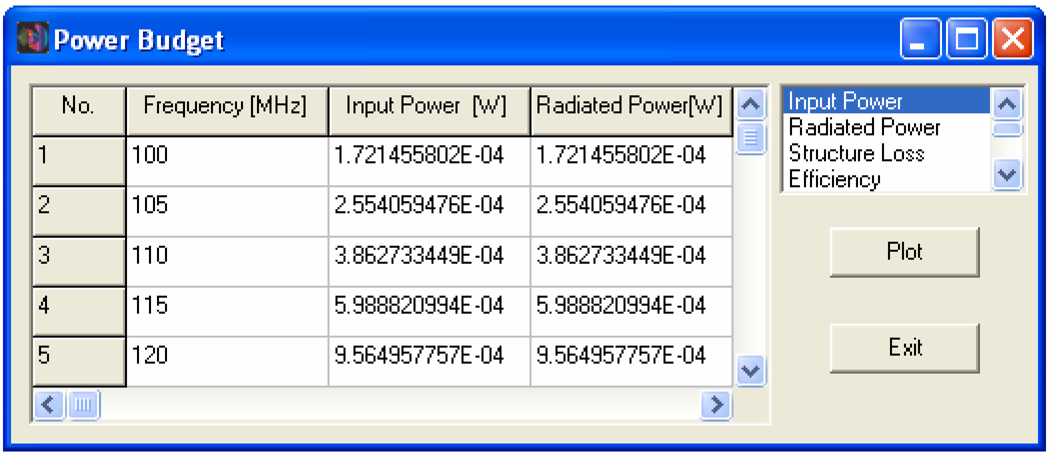


Fig. 13.21: The Power Budget dialog box.

13.12 Radar Cross Section

Choose Results/Power Budget/RCS… in the main menu to display the Radar Cross Section dialog box, Fig. 13.22. This dialog box shows a list of the following items versus frequency when a plane wave was used as the excitation of the structure:

- The column **RCS [m^2]** shows the Radar Cross Section in square meters.
- The column **RCS [$lambda^2$]** shows the Radar Cross Section in square wavelengths.
- The column **RCS [dB]** shows the Radar Cross Section in decibels with a square wavelength taken as the reference value.
- The column **Radiated Power** shows the total radiated or scattered power from the structure.
- The column **Structure Loss** shows the total consumed power or ohmic losses in the structure.
- The column **Pav** is the average power density scattered from the structure. This value is computed averaging the scattered power density over all directions in space.
- The column **Pmax** is the maximum value of the scattered power density.
- The columns **Theta (max)** and **Phi (max)** are the zenith and azimuth angles, respectively, in the direction of maximum radiation.

Select an item from the list and press the Plot button for plotting the selected item as a function of frequency.

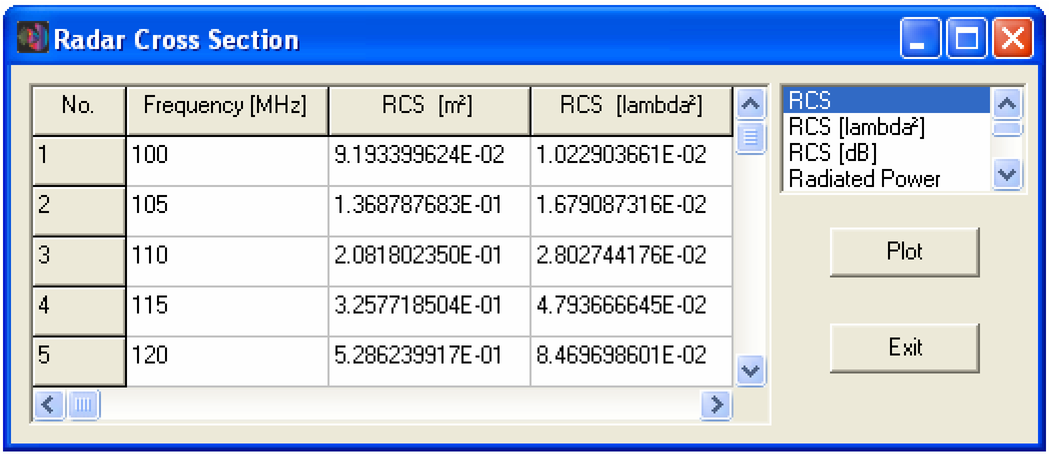


Fig. 13.22: The Radar Cross Section dialog box.

Important Information

The Radar Cross Section, the radiated power and the average power density are meaningful quantities only if the Theta and Phi angles in the Far-Field page of the Configuration dialog box are varying in the following ranges:

If the environment is **free space** (there is no ground plane):

$0 \leq \text{Theta} \leq 180$ [degrees]

and

$0 \leq \text{Phi} \leq 360$ [degrees]

If the environment has a **ground plane**:

$0 \leq \text{Theta} \leq 90$ [degrees]

and

$0 \leq \text{Phi} \leq 360$ [degrees]

This is because the average power density must be computed averaging the power density or Poynting vector by taking into account all directions in free space. If there is a ground plane, directions must be considered in half-space.

13.13 Plotting Near-Field Patterns

The computed near electric field can be shown as a 3D color map plot by choosing the Results/Plot Near E-Field Pattern/3D Plot command in the main menu. This command executes the AN-3D Pattern® program, Fig. 13.23. Besides, the near magnetic field can be plotted by selecting Results/Plot Near H-Field Pattern/3D Plot in the main menu.

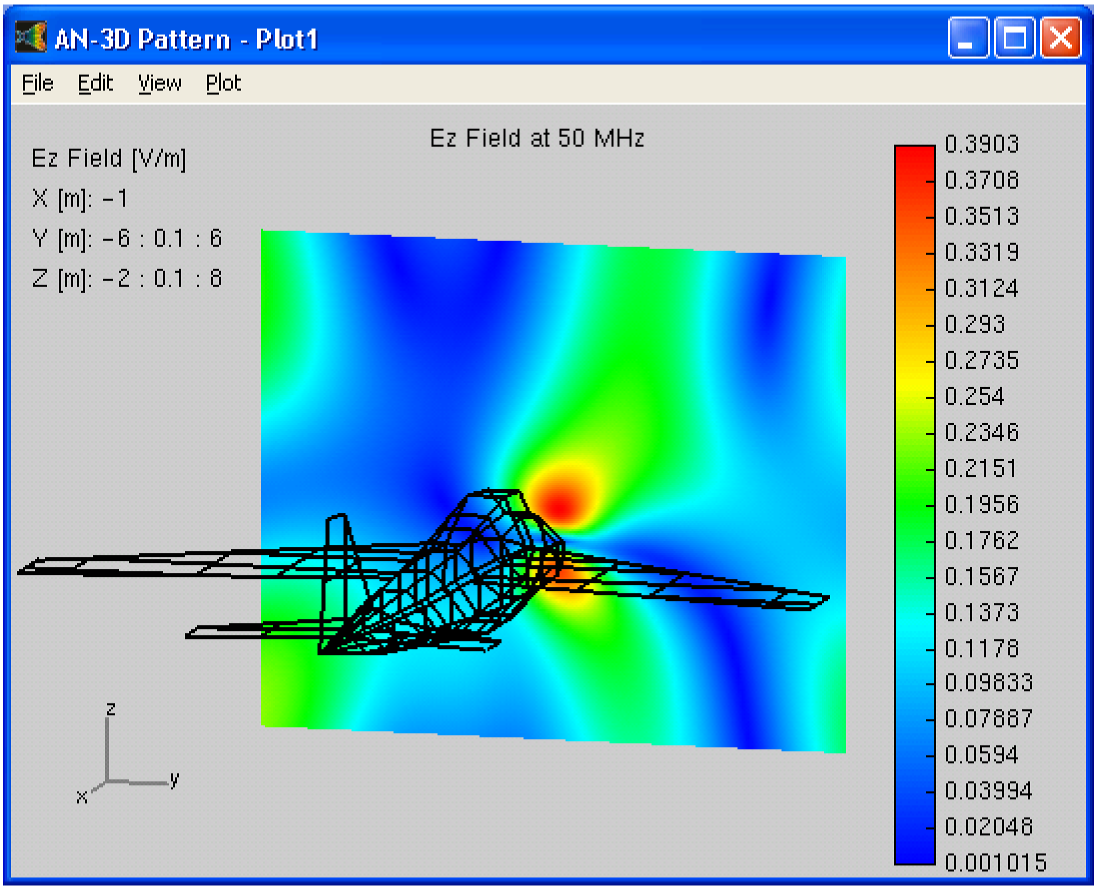


Fig. 13.23: Near-field 3D plot shown by AN-3D Pattern®.

Near-field 3D plots will be shown according to the type of coordinate system that was chosen in the Near-Field page of the Configuration dialog box: Cartesian, Cylindrical or Spherical.

If near-fields were calculated for several frequencies, a dialog box asking for a fixed frequency will be shown before plotting the near-field pattern.

The near electric field can also be shown as a 2D rectangular plot by choosing the Results/Plot Near E-Field Pattern/2D Plot command in the main menu. Besides, the near magnetic field can be plotted by selecting Results/Plot Near H-Field Pattern/2D Plot in the main menu. Then, the AN-XY Chart® program is executed, where the total rms electric or magnetic field is plotted in a 2D diagram, Fig. 13.24.

If Cartesian coordinates have been selected in the Near-Field page of the Configuration dialog box, the E_x, E_y and E_z electric field components and the H_x, H_y and H_z magnetic field components willbe calculated in a rectangular grid of points in space with coordinates (x,y,z).

If Cylindrical coordinates have been selected in the Near-Field page of the Configuration dialog box, the E_r, E_ϕ and E_z electric field components and the H_r, H_ϕ and H_z magnetic field components will be calculated in a cylindrical grid of points in space with coordinates (r,ϕ,z).

If Spherical coordinates have been selected in the Near-Field page of the Configuration dialog box, the E_r, E_θ and E_ϕ electric field components and the H_r, H_θ and H_ϕ magnetic field components will be calculated in a spherical grid of points in space with coordinates (r,θ,ϕ).

Any near-field component can be plotted by choosing between the options under Plot in the AN-XY Chart® main menu.

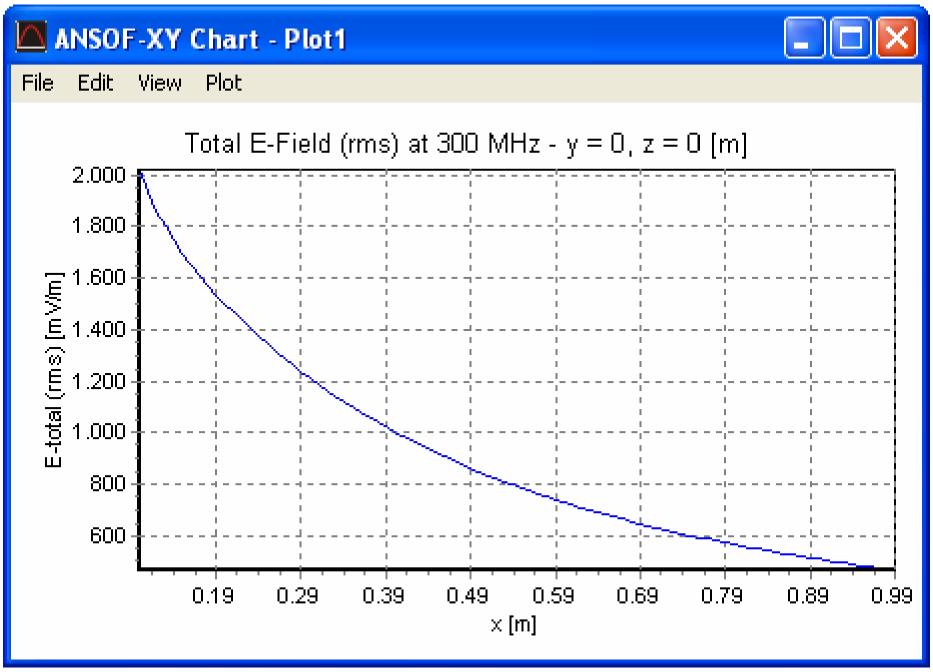


Fig. 13.24: Near electric field plotted by the AN-XY Chart® program as a function of the x-coordinate.

13.14 Plotting Near-Field Spectra

Near-field frequency spectra are obtained when a simulation is performed by specifying a list of frequencies or a frequency sweep. For each frequency, the near-field is calculated at the several points specified in the Near-Field page of the Configuration dialog box. Therefore, in order to plot the computed near-field versus frequency, a fixed point in space must be chosen.

If Cartesian coordinates are selected in the Near-Field page of the Configuration dialog box, the E_x, E_y and E_z electric field components and the H_x, H_y and H_z magnetic field components are calculated in a rectangular grid of points (x,y,z), so fixed x, y, and z coordinates must be chosen.

If Cylindrical coordinates are selected in the Near-Field page of the Configuration dialog box, the E_r, E_ϕ and E_z electric field components and the H_r, H_ϕ and H_z magnetic field components are calculated in a cylindrical grid of points (r,ϕ,z), so fixed r, ϕ, and z coordinates must be chosen.

If Spherical coordinates are selected in the Near-Field page of the Configuration dialog box, the E_r, E_θ and E_ϕ electric field components and the H_r, H_θ and H_ϕ magnetic field components are calculated in a spherical grid of points (r,θ,ϕ), so fixed r, θ and ϕ coordinates must be chosen.

The near-field spectrum can be plotted via Results/Plot Near E-Field Spectrum for the electric field and Results/Plot Near H-Field Spectrum for the magnetic field, both commands in the main menu. These commands display the Select Near-Field Point dialog box, where the fixed point (x,y,z), (r,ϕ,z) or (r,θ,ϕ) can be selected, Figs. 13.25. After pressing the OK button, the AN-XY Chart® program will show the frequency spectrum of the total near electric or magnetic field, Fig. 13.26.

The field components can be plotted in amplitude, phase, real and imaginary parts by choosing these options under Plot in the AN-XY Chart® main menu.

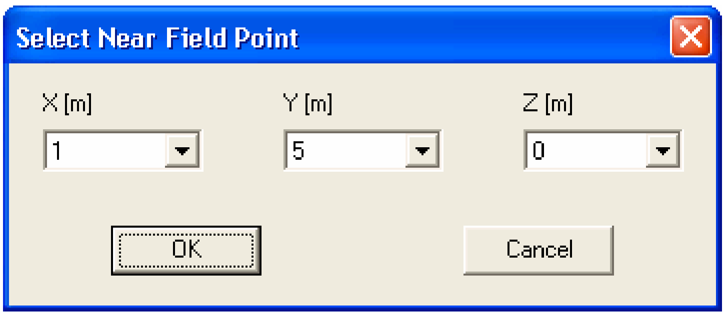


Fig. 13.25: Select Near-Field Point dialog box for selecting a fixed point (X,Y,Z) when Cartesian coordinates are used.

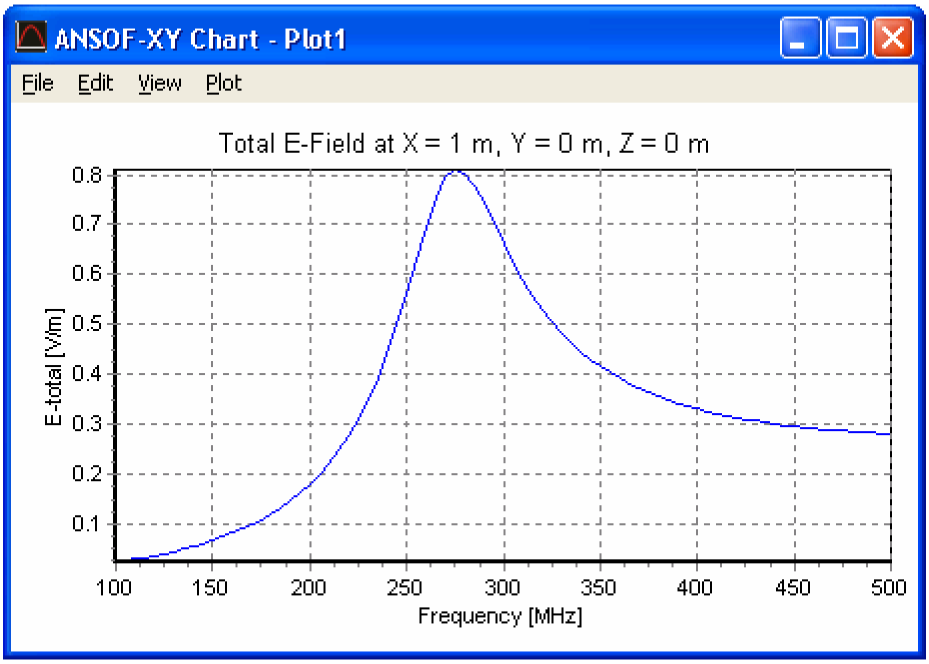


Fig. 13.26: Near E-field spectrum plotted by the AN-XY Chart® program.

14. Step-by-Step Examples

14.1 Simulation of a Cylindrical Antenna

A straight wire with a voltage source at its center can simulate a center-fed cylindrical antenna. Following the steps listed below can perform the simulation.

Step 1: Choose Edit/Preferences in the main menu for selecting suitable units for frequencies and lengths. In this case, frequencies will be measured in MHz and length in mm. Then, go to Simulate/Configure... in the main menu. In the Frequency page of the Configuration dialog box choose Sweep and fill the Frequency Sweep box as shown in Fig. 14.1.

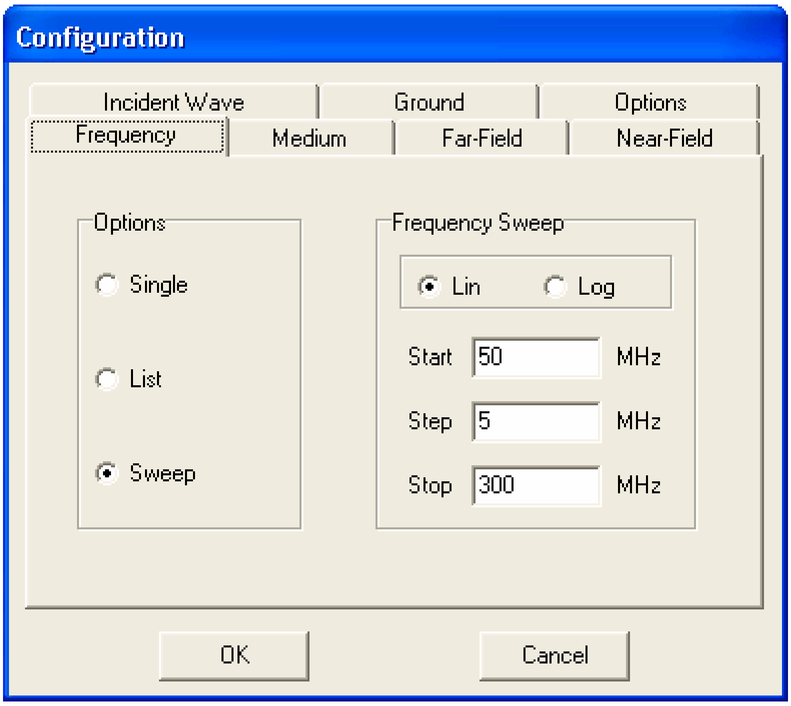


Fig. 14.1: The Frequency page in the Configuration dialog box. The computation will be performed at the frequencies: 50, 55,... ,295, 300 MHz.

Step 2: Choose Draw/Wire/Line in the main menu. The Draw dialog box for the Line will be shown. Fill the Line and Attributes pages as shown in Figs. 14.2 and 14.3. A straight wire with 17 segments will be drawn.

Clicking with the right mouse button on the wire shows a pop-up menu, where the Source/Load command can be selected. Follow the procedure described in Section 8.3 and put a voltage source in segment number 9, i.e. at the middle point of the wire. The source voltage is 1 (0°) V. The center-fed cylindrical antenna on the workspace is depicted in Fig. 14.4.

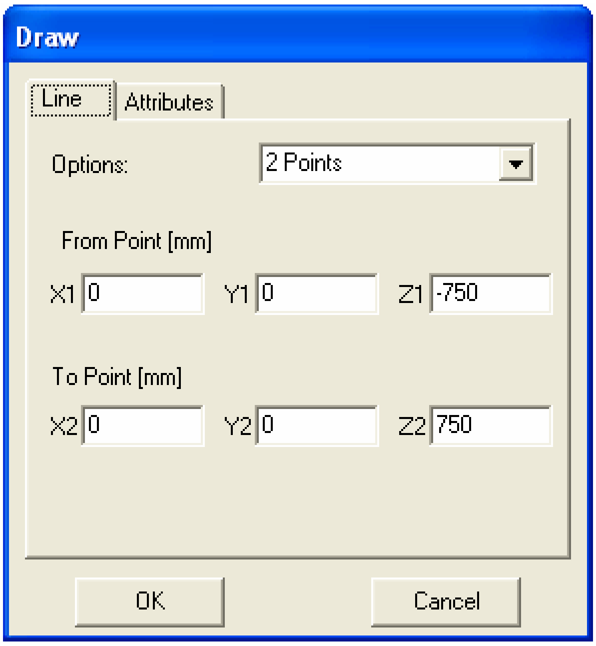


Fig. 14.2: The Line page in the Draw dialog box. The straight wire will be drawn starting from the point (0,0,-750) [mm] and ending at the point (0,0,750) [mm]. Thus, it is on the z-axis and is 1500 mm long, corresponding to half-wavelength at 100 MHz. Press F3 to view the main axes.

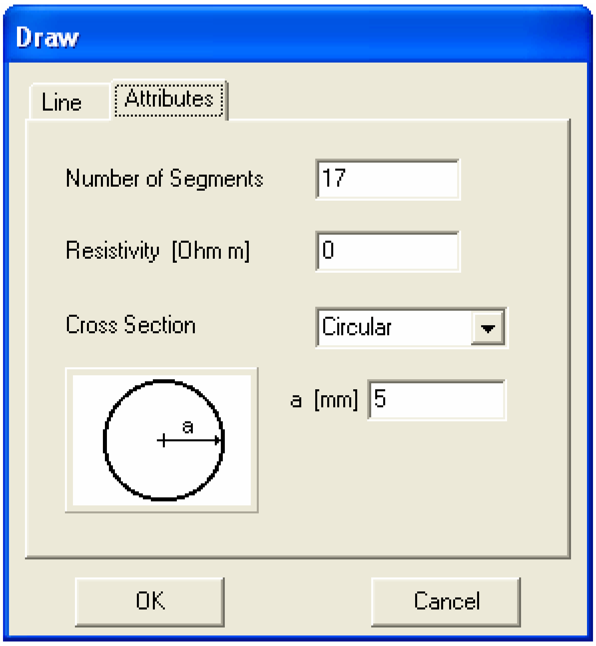


Fig. 14.3: The Attributes page in the Draw dialog box. The wire will be divided into 17 segments, its resistivity is set to zero (perfect electric conductor) and it has a circular cross-section with the radius set to 5 mm.

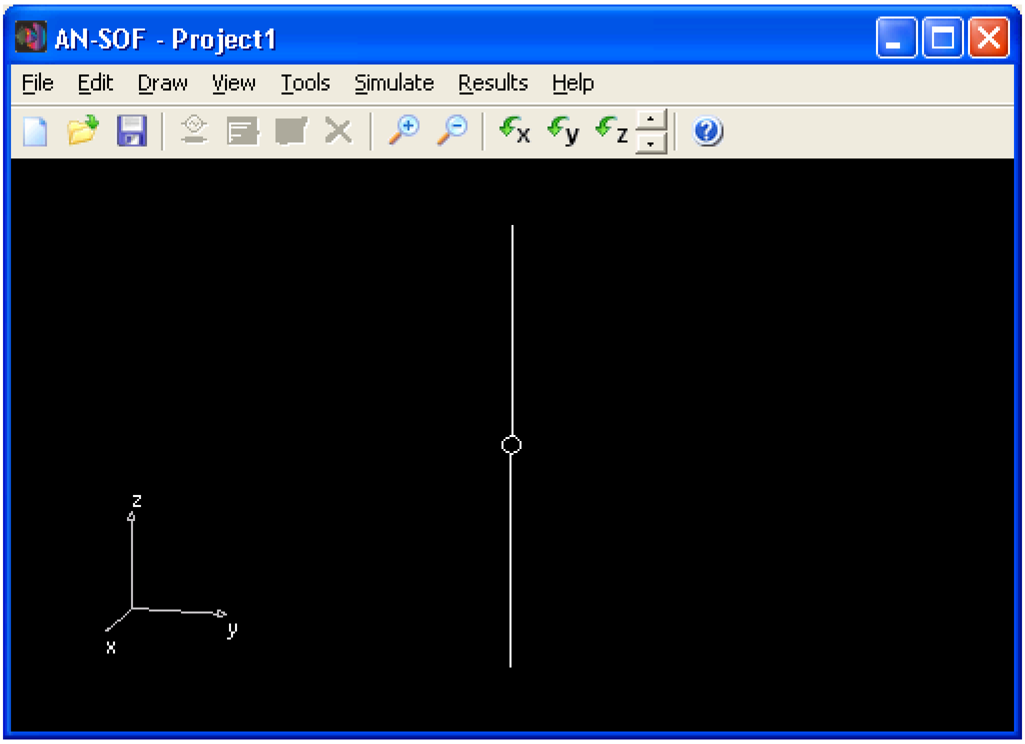


Fig. 14.4: Cylindircal antenna in the AN-SOF® workspace.

Step 3: Press Simulate/Run Currents in the main menu. Once the simulation has finished press Simulate/Run Far-Field.

Step 4: Clicking on the wire with the right mouse button, and selecting Plot Currents in the pop-up menu can show a plot of the current distribution. Follow the procedures described in Chapter 13 for visualizing other parameters of interest.

As an example, the current distribution in amplitude and phase at 100 MHz, the input impedance vs. frequency, and the directivity and E-field patterns in a 3D diagram at 100 MHz are shown in the following figures.

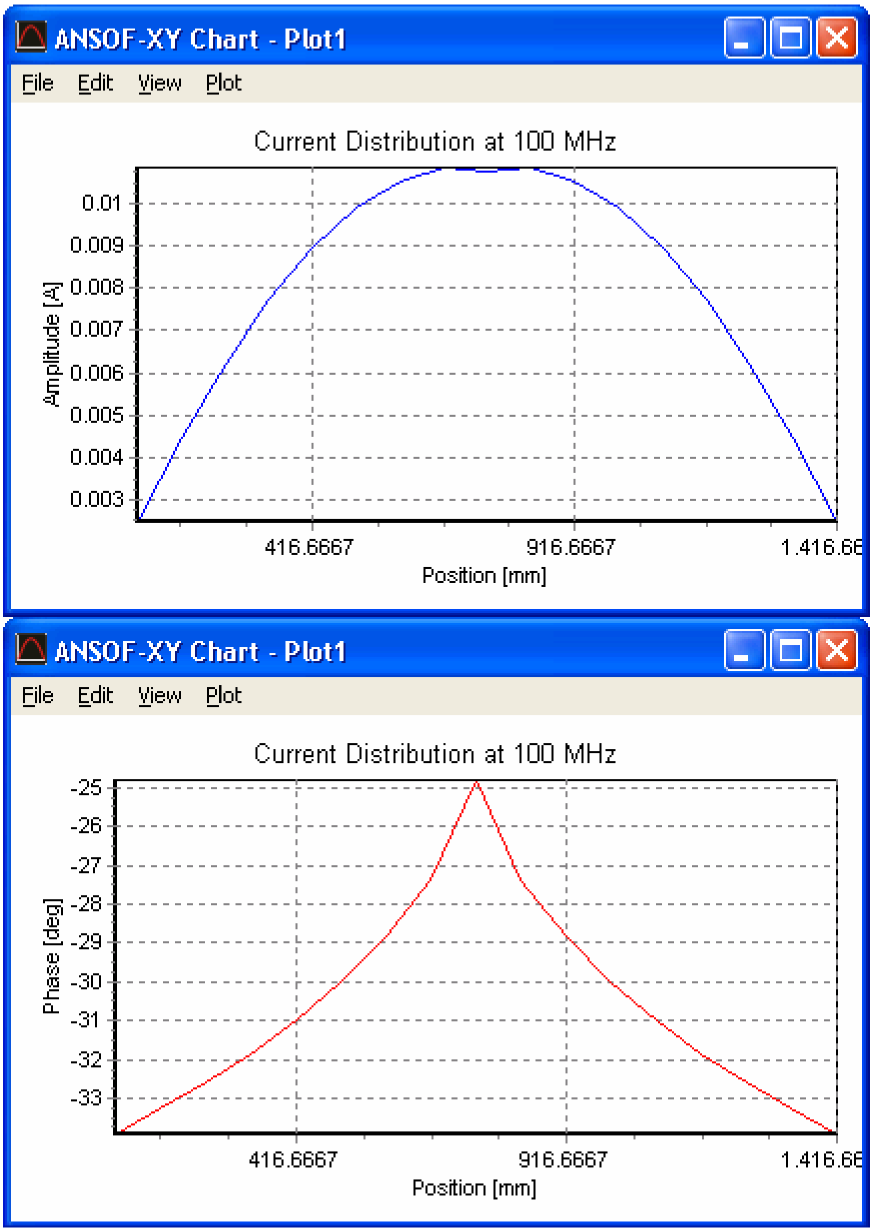


Fig. 14.5: Current distribution in amplitude and phase along the cylindrical antenna at 100 MHz.

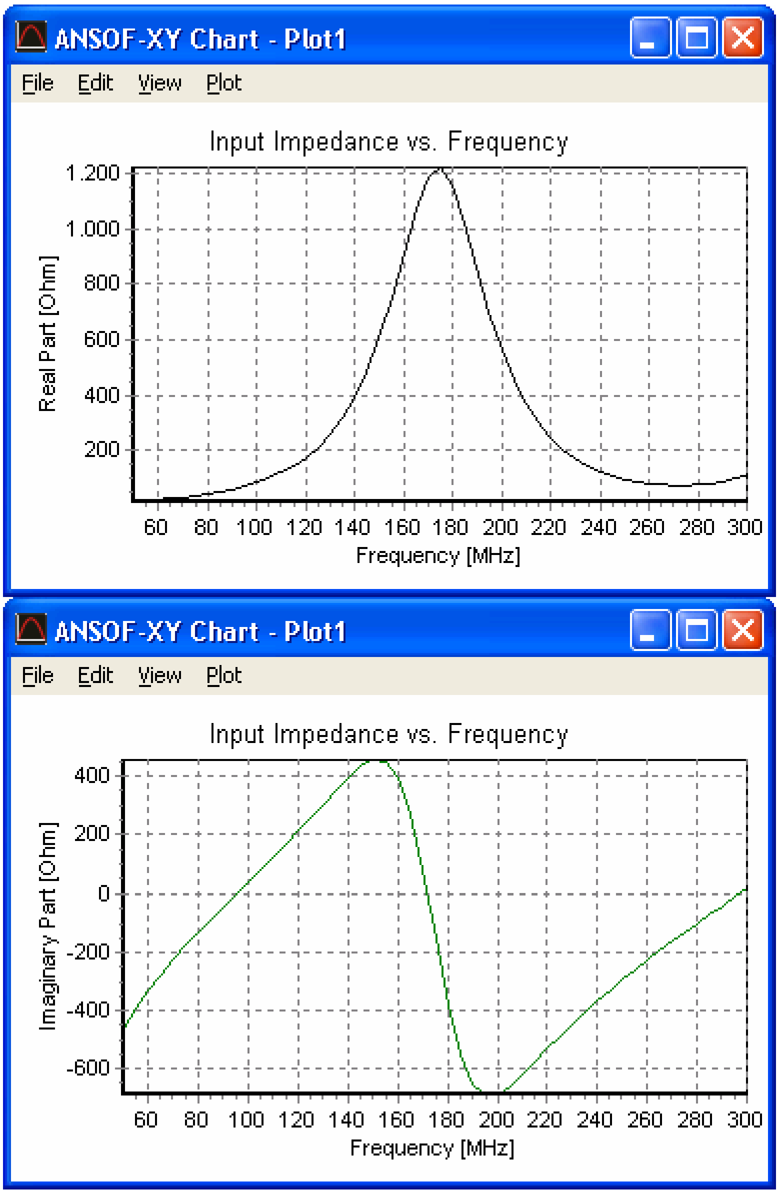
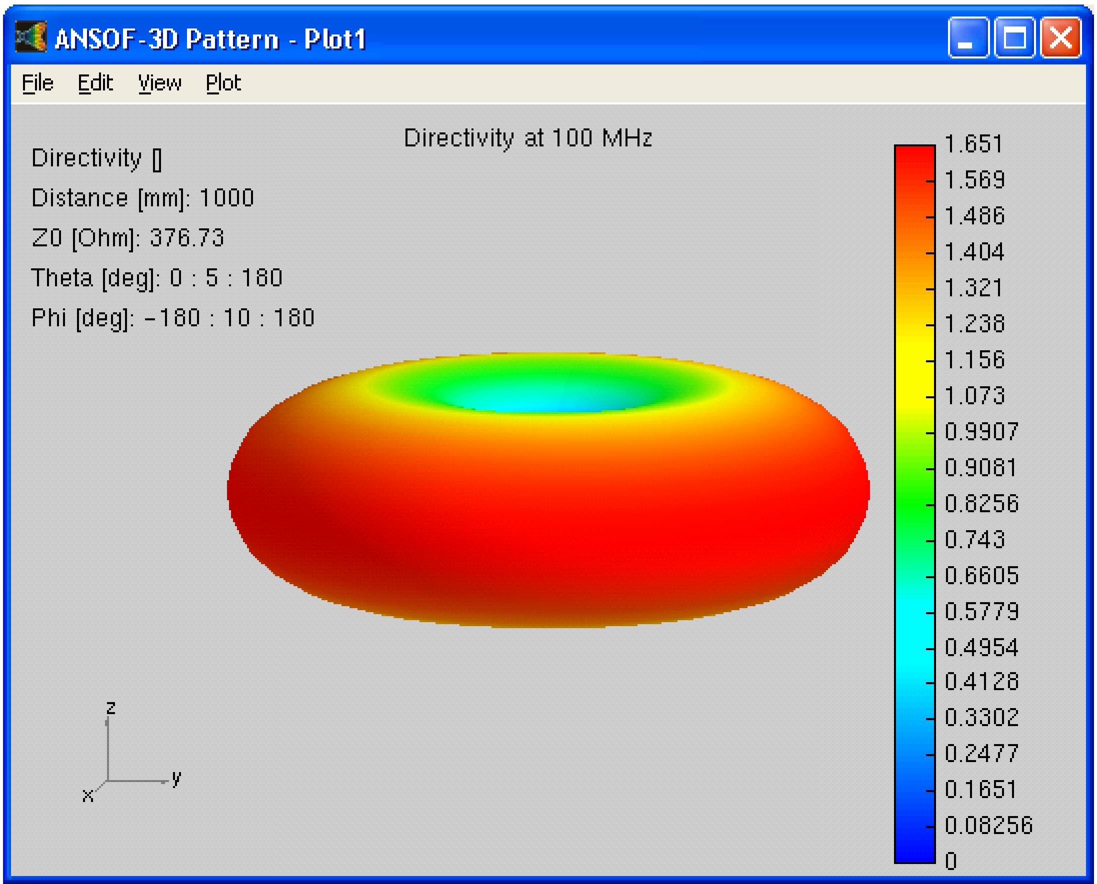


Fig. 14.6: Real and imaginary parts of the input impedance vs. frequency.



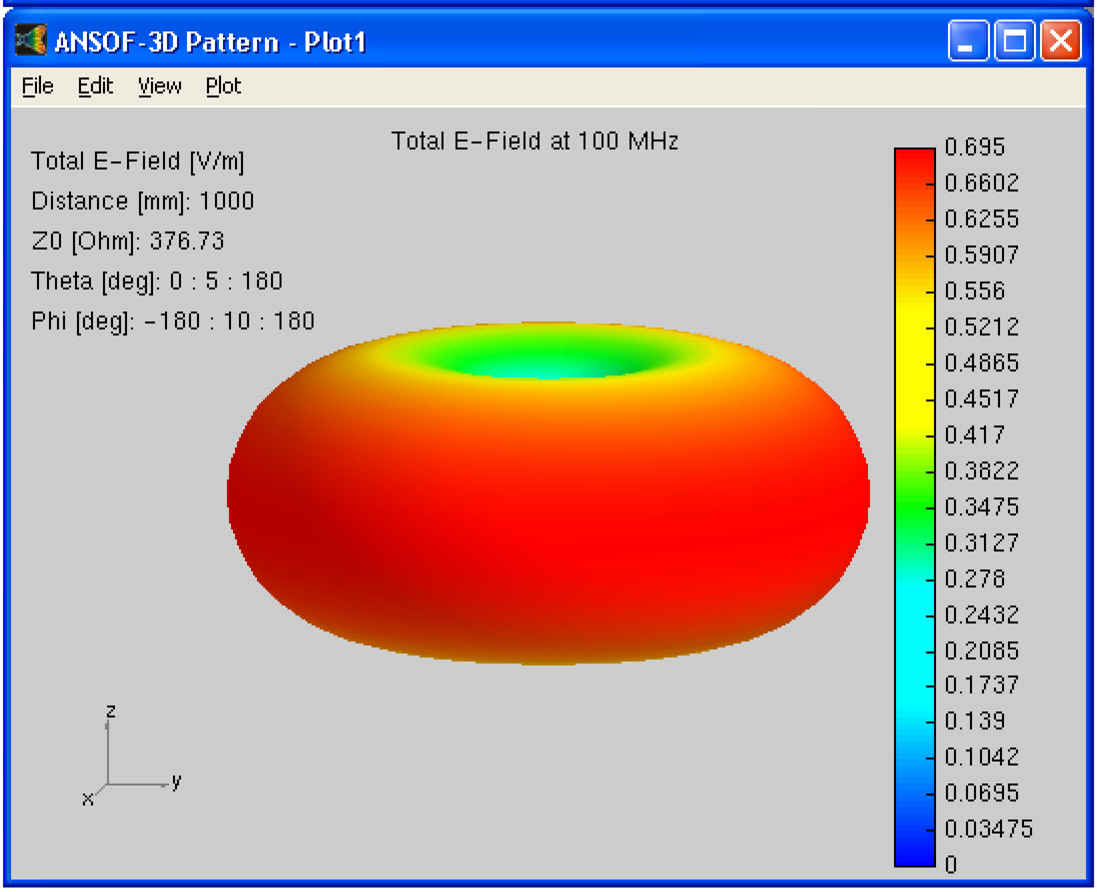


Fig. 14.7: Directivity and total E-field patterns at 100 MHz.

14.2 Simulation of a Transmission Line

A horizontal straight wire above a ground plane will simulate a transmission line. A sketch of the geometry definition is shown in Fig 14.8.

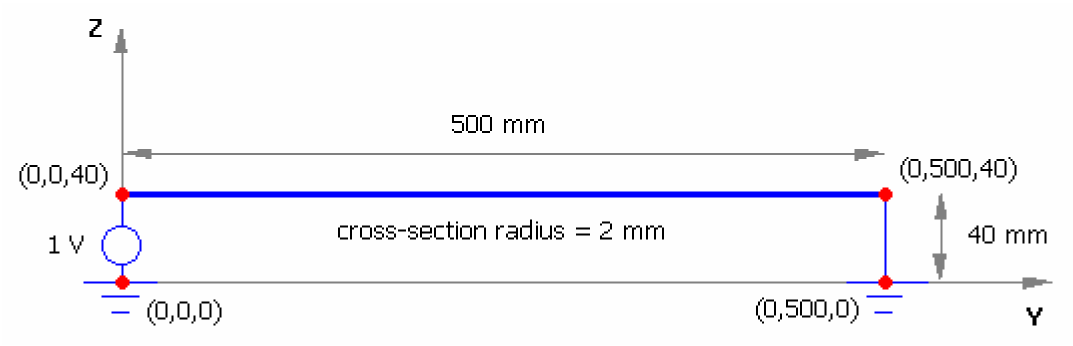


Fig. 14.8: Short-circuited transmission line.

The computation will be performed at 100 MHz. Following the steps listed below can perform the simulation.

Step 1: Go to Edit/Preferences in the main menu and select MHz and mm as frequency and length units, respectively. Choose Simulate/Configure... in the main menu. In the Frequency page of the Configuration dialog box select Single and fill the Single Frequency box as shown in Fig. 14.9. Then, choose the Ground page and fill it as shown in Fig 14.10.

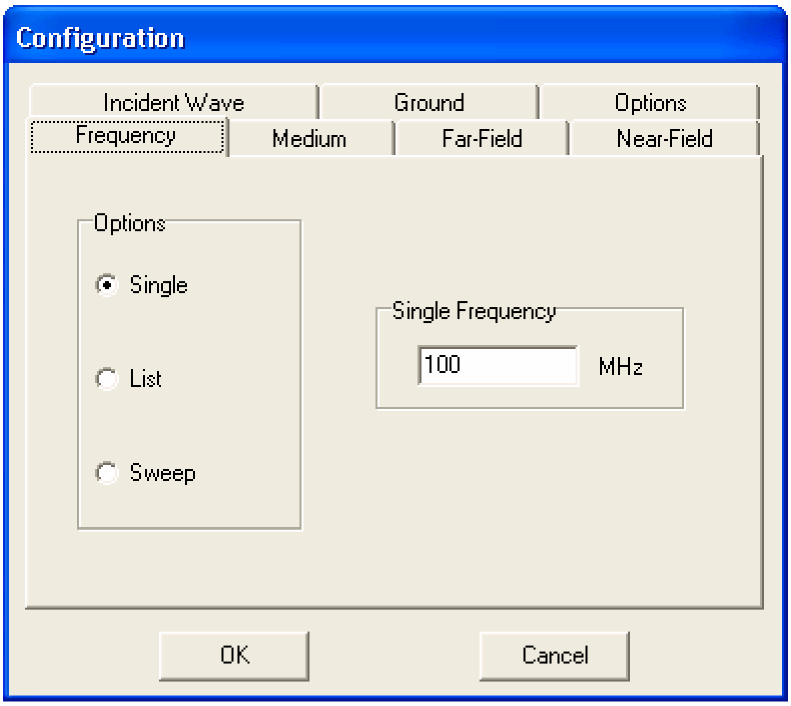


Fig. 14.9: The Frequency page in the Configuration dialog box.

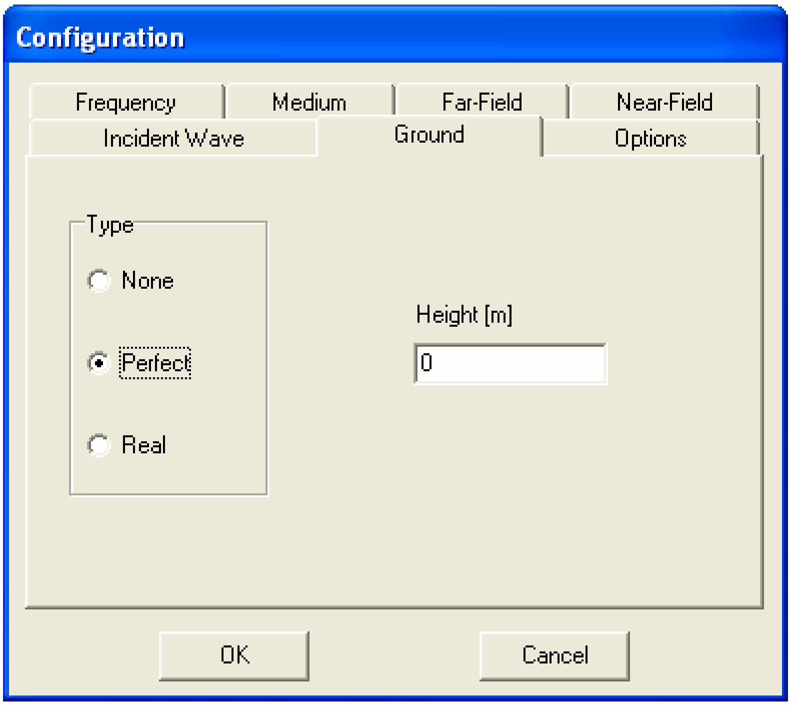


Fig. 14.10: The Ground page in the Configuration dialog box.

Step 2: Choose Draw/Wire/Line in the main menu. The Draw dialog box for the Line will be shown. Follow the procedure described in Section 5.1 for drawing linear wires to replicate the geometry in Fig. 14.8. The horizontal wire has 40 segments and the vertical short wires have 1 segment. The electrical connections at the points (0,0,40) [mm] and (0,500,40) [mm] are done automatically as well as the ground connections. The transmission line will be shown in the workspace as it is depicted in Fig. 14.11.

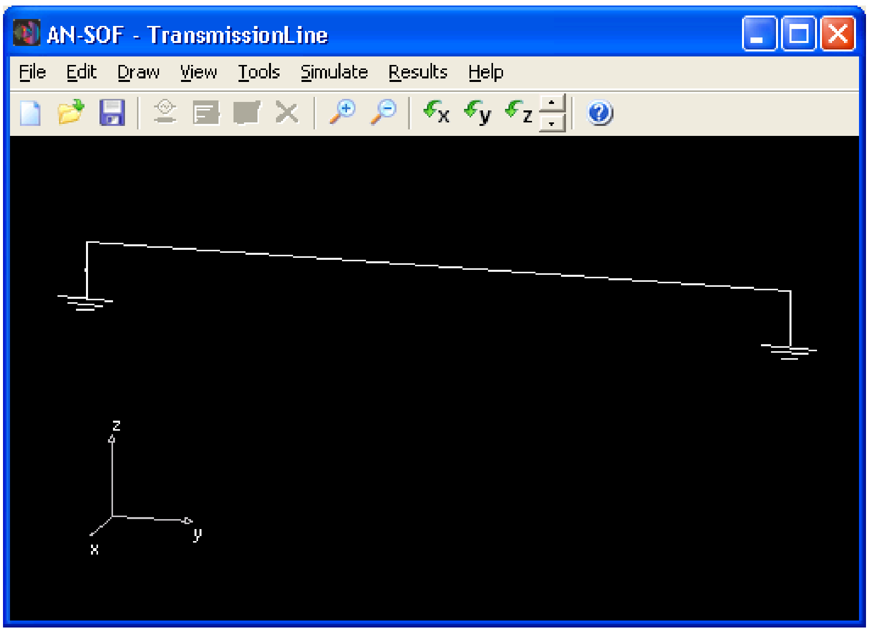


Fig. 14.11: Short-circuited transmission line in the AN-SOF® workspace.

Step 3: Press Simulate/Run Currents in the main menu.

Step 4: Follow the procedure described in Section 13.4 to obtain the input impedance of the short-circuited transmission line. Then, delete the right short wire shown in Fig. 14.11 to simulate an open-circuited transmission line and get its input impedance. These values are approximately:

- Z_{in} (short-circuited line) = j 510 Ohm.
- Z_{in} (open-circuited line) = -j 105 Ohm.

Then, the characteristic impedance of the line is given by:

$$Z_c = \sqrt{510 \times 105} = 231 \text{ Ohm}$$

On the other hand, the relation for the characteristic impedance of a line above a ground plane is given by:

$$Z_c = 138 \log\left(\frac{2h}{a}\right) = 138 \log\left(\frac{2 \times 40}{2}\right) = 221 \text{ Ohm}$$

Where “a” is the wire cross-section radius and “h” is the height above the ground plane. As can be seen from these results, the agreement is quite good.

Step 5: Go back to the short-circuited transmission line in Fig. 14.11 by pressing Edit/Undo in the main menu. Choose Simulate/Configure... in the main menu and fill the Frequency page as shown in Fig. 14.12.

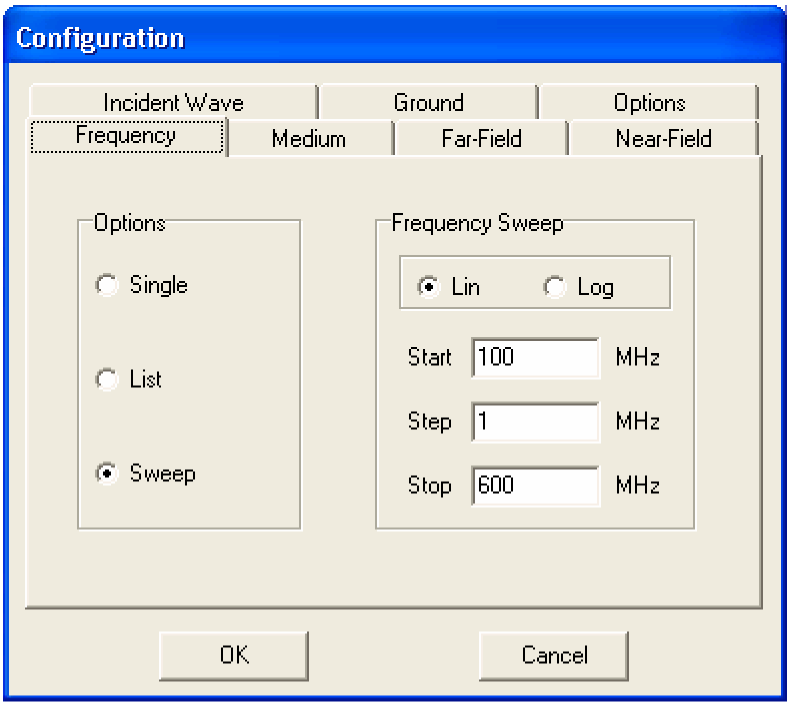


Fig. 14.12: Frequency sweep from 100 to 600 MHz in steps of 1 MHz.

Step 6: Press Simulate/Run Currents in the main menu.

Step 7: Follow the procedure described in Section 13.5 for showing Smith charts. The input impedance curve will be plotted by the AN-Smith® program, Fig. 14.13. The input admittance can be seen by pressing Plot/Admittance in the AN-Smith® main menu.

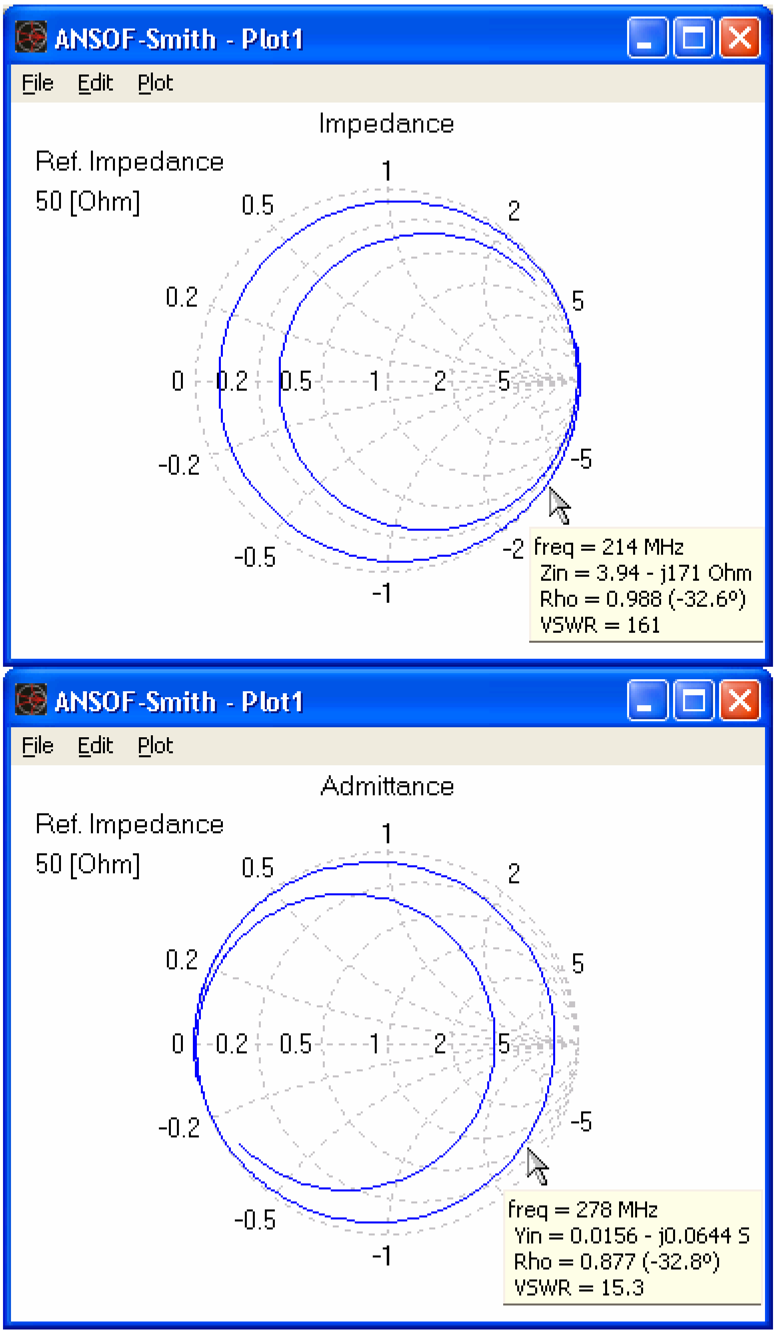


Fig. 14.13: Input impedance and admittance plotted in a Smith chart.

The reference impedance for reflection coefficients (Rho) and VSWR is set to 50 Ohm. This reference can be changed going to Simulate/Configure… in the AN-SOF® main menu and choosing the Options page.

14.3 Simulation of an RLC Circuit

A sketch of the geometry definition for the RLC circuit is shown in Fig 14.14. Resonance is obtained at the frequency:

$$f_o = \frac{1}{2\pi\sqrt{LC}} = 796 \ \text{Hz}$$

This frequency will be verified by a frequency sweep computation. Following the steps listed below can perform the simulation.

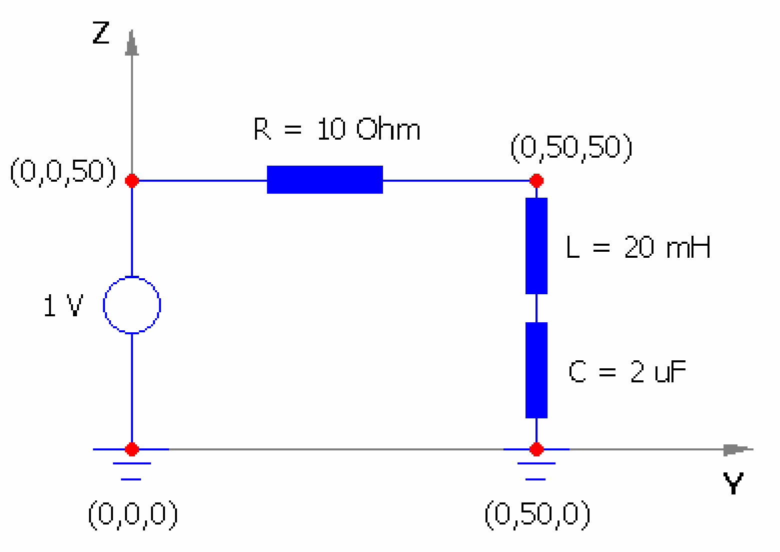


Fig. 14.14: Geometry definition for the RLC circuit.

Step 1: Go to Edit/Preferences in the main menu and select the Hz , mm, mH and uF as the units for frequency, length, inductance and capacitance, respectively. Then, choose Simulate/Configure... in the main menu. In the Frequency page of the Configuration dialog box choose "Sweep" and fill the "Frequency Sweep" box as shown in Fig. 14.15. Then, choose the Ground page and fill it as shown in Fig 14.16.

Step 2: Choose Draw/Wire/Line in the main menu. The Draw dialog box for the Line will be shown. Follow the procedure described in Section 5.1 for drawing linear wires to replicate the geometry in Fig. 14.14. The left vertical wire has 1 segment, the horizontal wire has 1 segment and the right vertical wire has 3 segments. The electrical connections at the points (0,0,50) [mm] and (0,50,50) [mm] are done automatically as well as the ground connections. Follow the procedure described in Section 8.5 for adding the load impedances shown in Fig. 14.14. The RLC circuit in the workspace is depicted in Fig. 14.17.

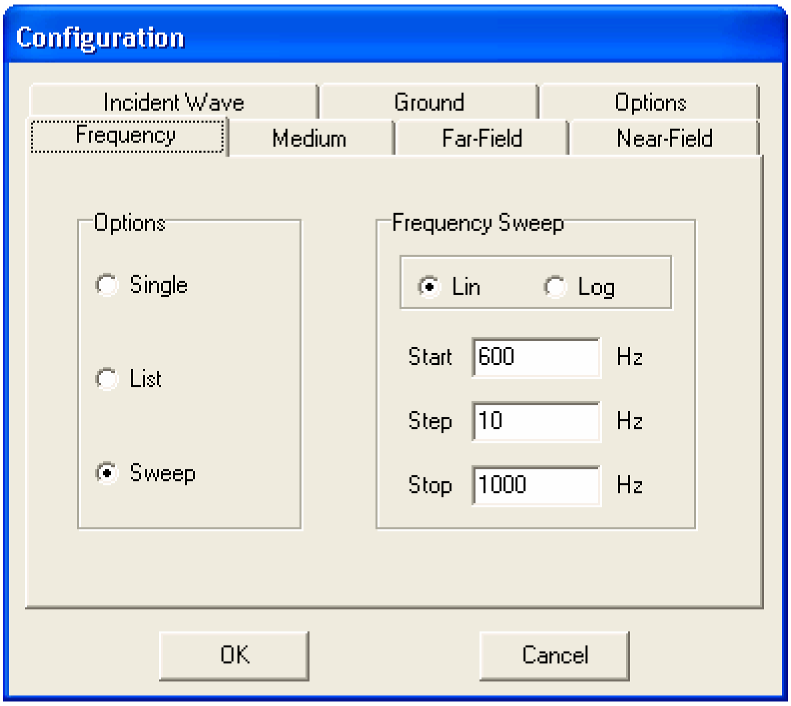


Fig. 14.15: The Frequency page in the Configuration dialog box. The computation is performed at the frequencies: 600, 610,... ,990, 1000 Hz.

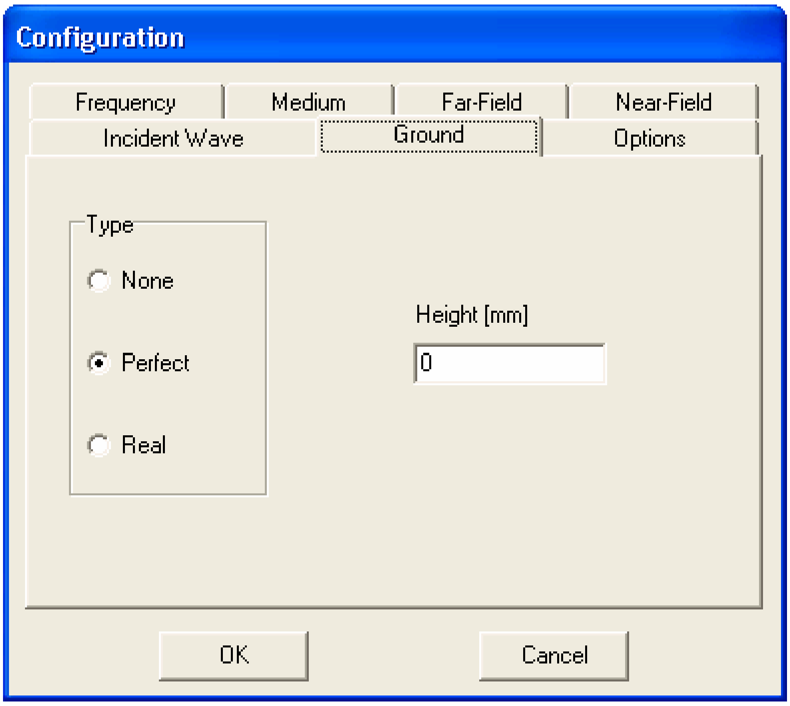


Fig. 14.16: The Ground page in the Configuration dialog box.

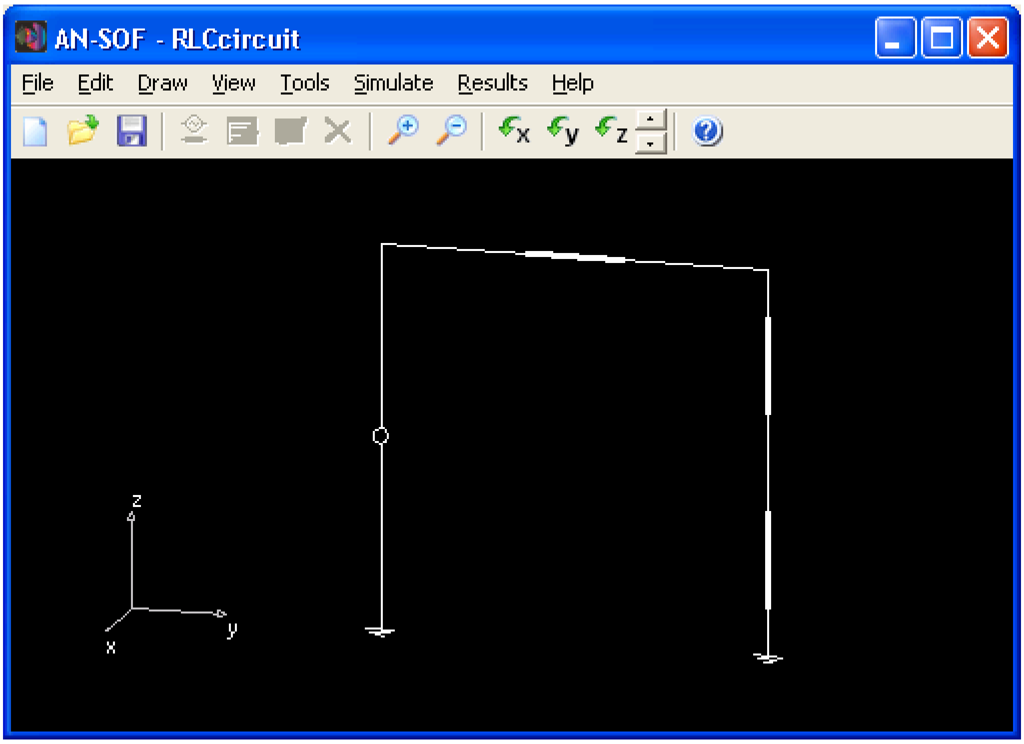


Fig. 14.17: RLC circuit in the AN-SOF® workspace.

Step 3: Press Simulate/Run Currents in the main menu.

Step 4: Select any of the three wires composing the circuit and follow the procedure described in Section 13.3 for plotting the current on a wire segment as a function of frequency, Fig 14.18. Due to the circuit topology, the electric current must be the same in the three wires. As can be seen, resonance occurs at a frequency near to 800 Hz. In fact, the value 796 Hz can be verified by simulating at this single frequency.

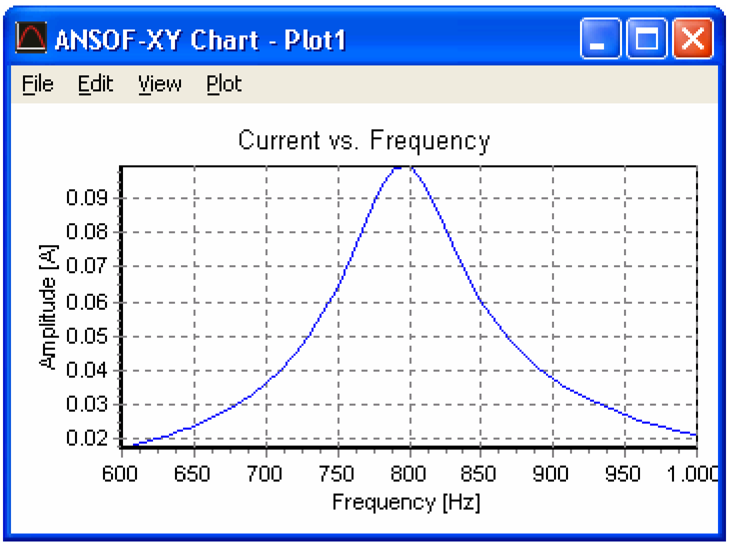


Fig. 14.18: Current amplitude vs. frequency in the RLC circuit.

14.4 Yagi-Uda Antenna

A Yagi-Uda antenna consisting of a driven element, one director and one reflector can be simulated by an array of three linear wires, as Fig. 14.19 shows, where the Cartesian coordinates of the wire ends are indicated in meters. The operating frequency is set to 300 MHz in the Configuration dialog box.

Follow the procedure described in the simulation of a cylindrical antenna for drawing each linear wire at a time. Then, connect a voltage generator at the middle point of the driven element. Each wire is divided into 15 segments and wire radius is 5 mm. The angular ranges for calculating the far-field are Theta = 0:2.5:180 deg and Phi = 0:5:360 deg.

Figure 14.20 shows the Power Budget dialog box, where a peak directivity of about 7.8 (or 8.9 dBi) is obtained. This can also be seen in the directivity pattern of the Yagi-Uda array shown in Fig. 14.21.

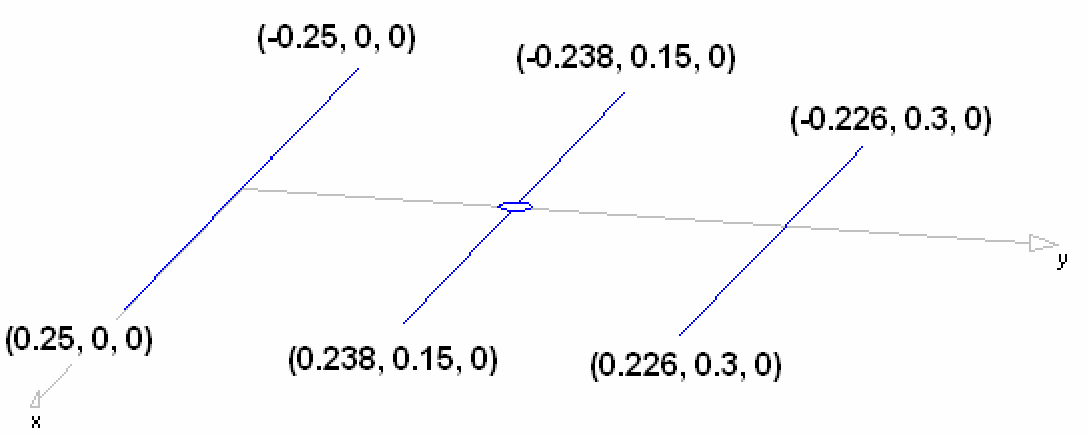


Fig. 14.19: Geometry definition for the Yagi-Uda array.

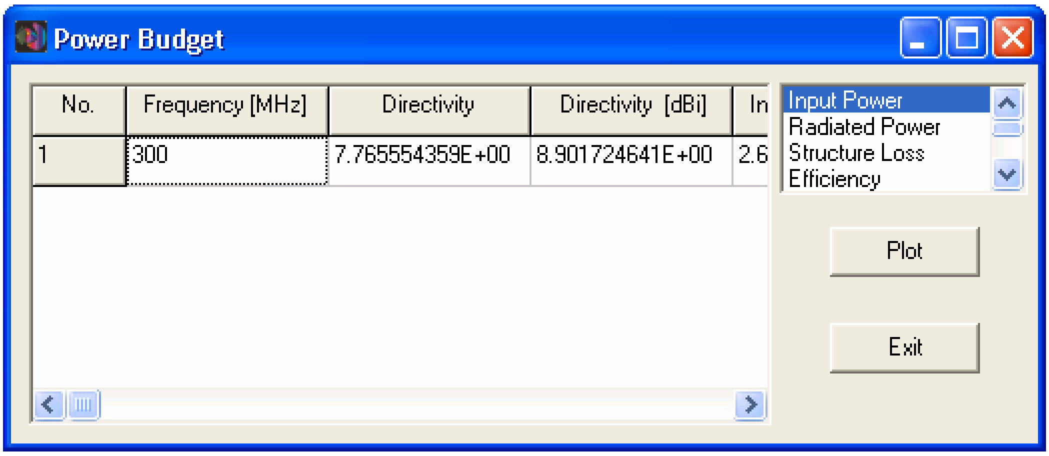


Fig. 14.20: Power Budget dialog box, where a peak directivity of 7.53 is obtained for the Yagi-Uda array.

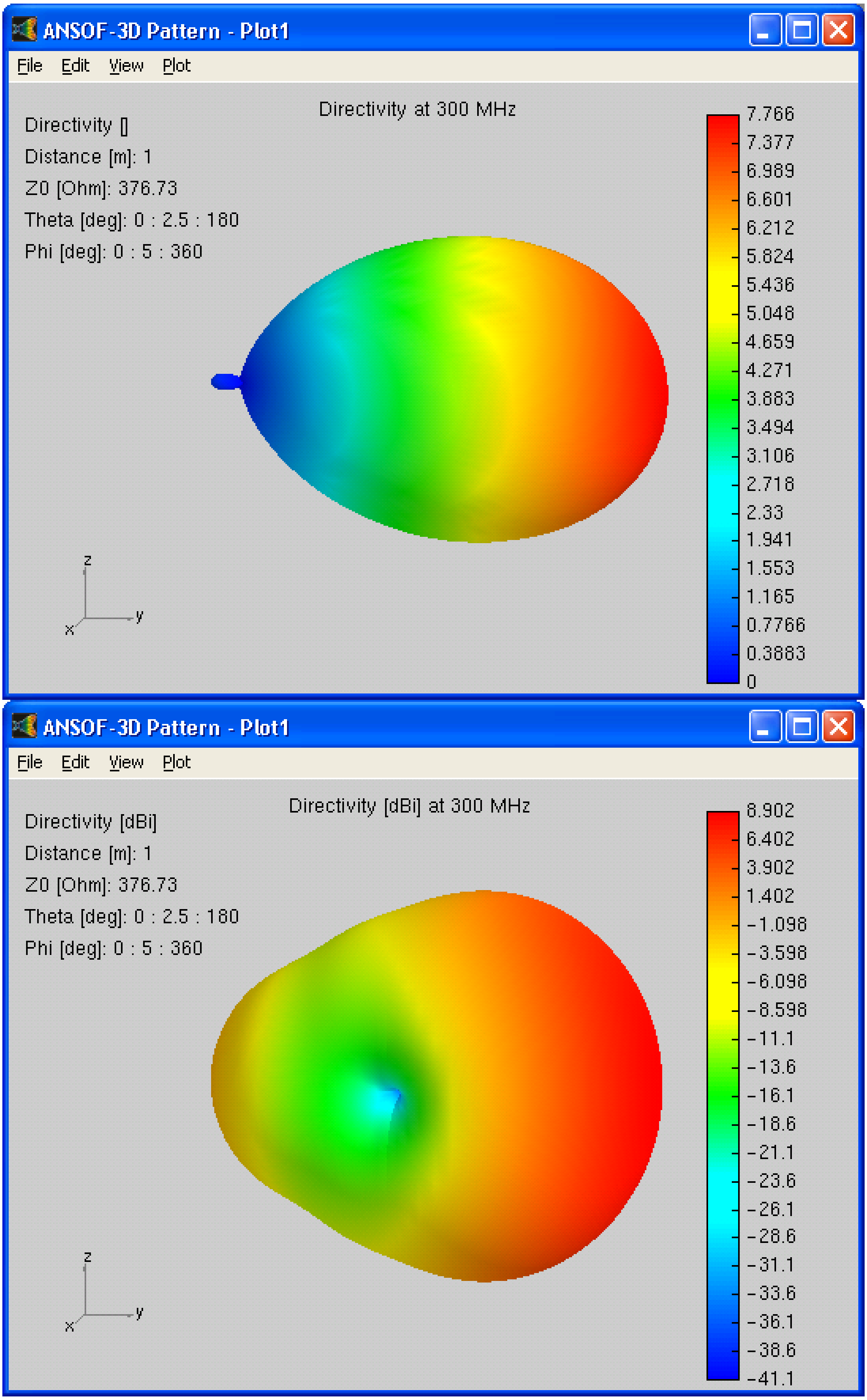


Fig. 14.21: Directivity pattern for the Yagi-Uda array of Fig. 14.19.

15. Shortcut Keys

Pressing ALT with the underlined letter of a push button will execute the command associated with the button.

The following keys and associated actions are available:

Key	Action
Home	Return the structure to the initial view
Left Arrow ←	Select a wire
Right Arrow →	Select a wire
+	Zoom in
-	Zoom out
Q	Rotate around +X axis
A	Rotate around -X axis
W	Rotate around +Y axis
S	Rotate around -Y axis
E	Rotate around +Z axis
D	Rotate around -Z axis
Ctrl+N	Create a new project
Ctrl+O	Open a file
Ctrl+S	Save the project
Ctrl+Q	Exit AN-SOF®
Ins	Display the Source/Load toolbar
Ctrl+Ins	Modify the selected wire
Del	Delete the selected wire or group of wires
Ctrl+W	Show properties of the selected wire
Ctrl+A	Display the Axes dialog box
Ctrl+C	Display the Configuration dialog box
F1	Help
F3	Show Main/Small axes
F5	Run ALL
F6	Run currents and far-field
F7	Run currents and near-field
F8	Plot 3D current distribution
Ctrl+F8	Plot 2D currents in a selected wire
F9	Plot 3D far-field pattern
Ctrl+F9	Plot far-field in a 2D polar diagram
F10	Plot 3D near electric field
Ctrl+F10	Plot 2D near electric field
F11	Plot 3D near magnetic field
Ctrl+F11	Plot 2D near magnetic field
ESC	Unselect a wire

16. File Formats

When a project is saved AN-SOF® will save different files. These files will have the same name that the user gave to the project, but with a unique extension referring to the contents in the file.

The AN-SOF® file types are:

File type	Description
***.emm**	Main file of the program.
***.wre**	File containing geometrical data.
***.cur**	File containing currents flowing on wires.
***.phi**	E-phi component of the far-field.
***.the**	E-theta component of the far-field.
***.pwr**	File containing information about the radiation patterns.
***.nef**	Near electric field
***.nhf**	Near magnetic field
***.ngf**	Numerical green’s function
***.txt**	Text file written by the user

17. Getting Help

Context-sensitive Help is available from nearly every portion of the AN-SOF® program system.

To get context-sensitive Help place the mouse cursor on the button, menu, or item in a dialog box for which you want Help, and then press F1.

You can also get help in the Help menu (see Section 3.1 Main menu).

18. Background Theory

The **AN-SOF®** engine is written in C++ using double-precision arithmetic and has been developed to improve the accuracy in the modeling of wire antennas and general metallic structures.

The computer code is based on an Electric Field Integral Equation (EFIE) expressed in the frequency domain. The current distribution on thin-wire structures is computed by solving the EFIE using a Method of Moments (MoM) formulation with curved basis and testing functions, called the Conformal Method of Moments (CMoM). In this method, curved wires are modeled by means of conformal segments, which exactly follow the contour of the structure, instead of the traditional approximation based on straight wire segments. The linear approximation to the geometry can be a very inefficient method in terms of unknowns or computer memory. Nevertheless, by using curved segments, the number of unknown currents, simulation time and memory space can be greatly reduced, allowing for the solution of bigger problems.

Here is a brief explanation of the theoretical basis for the **AN-SOF®** program system.

18.1 Electric Field Integral Equation for Curved Wires

The current distribution on metallic surfaces with ideal conductivity can be found by solving an Electric Field Integral Equation (EFIE) expressed in the frequency domain:

$$\mathbf{n}\times\mathbf{E}_i(\mathbf{r}) = \mathbf{n}\times\frac{j}{\omega\varepsilon}\iint_S\left[k^2\mathbf{J}(\mathbf{r}')G(\mathbf{r},\mathbf{r}') + div'(\mathbf{J}(\mathbf{r}'))grad\,(G(\mathbf{r},\mathbf{r}'))\right]dS' \qquad (1)$$

where:

$\mathbf{E}_i$: Incident Electric Field on the surface S.
$\mathbf{n}$: unit vector at point $\mathbf{r}$ on the surface S.
k: wave number.
$\mathbf{J}$: unknown electric current density flowing on the surface.
G: Green's function, which in free space is given by:

$$G(\mathbf{r},\mathbf{r}') = \frac{e^{-jk|\mathbf{r}-\mathbf{r}'|}}{4\pi|\mathbf{r}-\mathbf{r}'|} \qquad (2)$$

The EFIE is an expression of a boundary condition on the surface, namely zero tangential electric field.

When we are dealing with a wire structure, the EFIE reduces to:

$$\mathbf{T}\cdot\mathbf{E}_i = \mathbf{T}\cdot\frac{j}{\omega\varepsilon}\int_\Gamma\left[k^2I(s')K(s,s')\mathbf{T}' + \frac{dI(s')}{ds'}grad\,(K(s,s'))\right]ds' \qquad (3)$$

where $\mathbf{T}$ is the tangential unit vector describing the contour of the curve Γ, $I(s)$ is the unknown electric current on the wire, and $K(s,s')$ is the integral equation kernel defined as:

$$K(s,s') = \frac{1}{4\pi^2}\int_0^{2\pi}\int_0^{2\pi}G(\mathbf{r},\mathbf{r}')d\phi'd\phi \qquad (4)$$

The EFIE is averaged about the wire circumference described by the angle ϕ, resulting in the EFIE (3) with the kernel (4). The current distribution $I(s)$ is then the average value of the current density $\mathbf{J}$ in the axial direction; the current in the ϕ direction is neglected. This is a good assumption provided that the wire radius is small with respect to the wavelength.

The wire axis Γ is defined by its parametric equations that can be written in the compact form:

$$\mathbf{r}(s) = x(s)\mathbf{i} + y(s)\mathbf{j} + z(s)\mathbf{k} \tag{5}$$

which points from the origin to any point on the wire. The parameter s varies over a real interval.

The tangent unit vector can be obtained from the first derivative of (5):

$$\frac{d\mathbf{r}(s)}{ds} = \frac{dx}{ds}\mathbf{i} + \frac{dy}{ds}\mathbf{j} + \frac{dz}{ds}\mathbf{k} \qquad \mathbf{T} = \frac{d\mathbf{r}}{ds}\left|\frac{d\mathbf{r}}{ds}\right|^{-1} \tag{6}$$

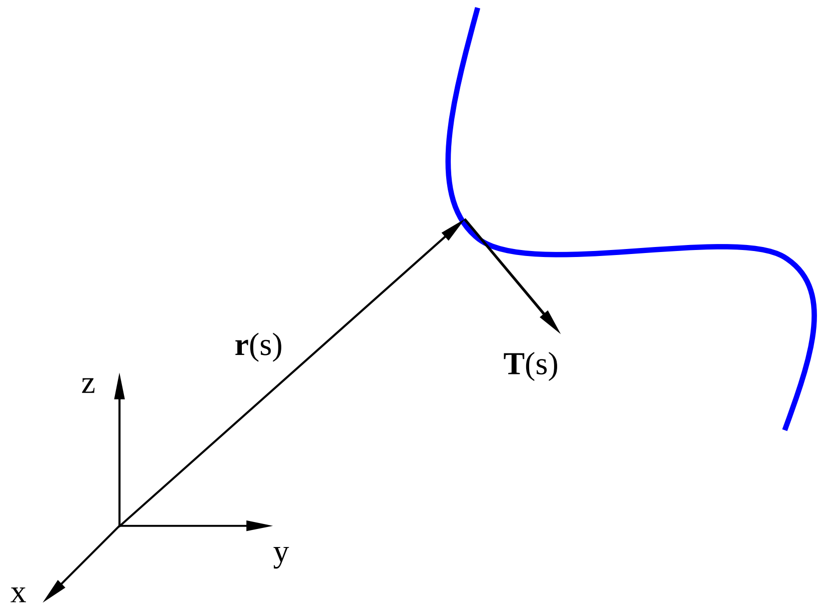


Fig. 18.1: Parametric description of a curved wire. The tangent unit vector is obtained from the first derivative of the position vector $\mathbf{r}(s)$.

This parametric description is the key for the accurate modeling of the wire structure. A straight wire approximation to the geometry produces a loss of geometrical information that can never be completely restored. However, this information is not lost if a parametric representation is used to describe the wire locus [3], [10], [11]. It is also possible to improve on the straight wire approximation by using quadratic segments to model the geometry [2].

Thus, the definition of a wire must include its parametric description and its first derivative if an exact representation of the geometry is required, as shown in Fig. 18.1.

The kernel (4) is approximated by the following generalized thin-wire approximation:

$$K(s,s') \cong \frac{e^{-jkR}}{4\pi R} \qquad R = \sqrt{\left|\mathbf{r}(s) - \mathbf{r}(s')\right|^2 + a^2} \tag{7}$$

where a is the wire radius.

When the observation point $\mathbf{r}(s)$ and the source point $\mathbf{r}(s')$ are both in the same straight wire, the distance R reduces to the usual thin-wire approximation:

$$R = \sqrt{(s - s')^2 + a^2} \tag{8}$$

Thus, the EFIE and its kernel are also valid for straight wires.

The existence of a PEC ground plane is modeled by means of image currents. This method can be easily implemented by adding an image term to the Green's function, resulting in an additional term for the kernel.

18.2 Curved Method of Moments

The Method of Moments (MoM) is a technique used to convert the EFIE into a system of linear equations that then can be solved by standard methods [1].

For simplicity, the integral (linear) operator in (3) will be denoted by L, then the EFIE takes the form:

$$L(I) = E_T \tag{9}$$

where E_T is the tangential component of the incident electric field.

The current distribution is approximated by a sum of N basis functions with unknown amplitudes I_n, giving:

$$I = \sum_n I_n F_n \tag{10}$$

With this expansion and using the linearity of the operator L, we can write:

$$\sum_n I_n L(F_n) = E_T \tag{11}$$

In order to obtain a set of N equations, the functional equation (11) is weighted with a set of N independent testing functions w_m, giving:

$$\sum_n I_n \int w_m L(F_n) ds = \int w_m E_T ds \tag{12}$$

where the integrals are calculated over the domain of L. Now we have as many independent equations as unknowns, so (12) can be written as:

$$[Z] \cdot [I] = [U] \tag{13}$$

where

$[Z]$: impedance matrix with dimension $N \times N$ and the elements $Z_{mn} = \int w_m L(F_n) ds$

$[I]$: current matrix with dimension $N \times 1$ and the elements I_n.

$[U]$: voltage matrix with dimension $N \times 1$ and the elements $U_m = \int w_m E_T ds$

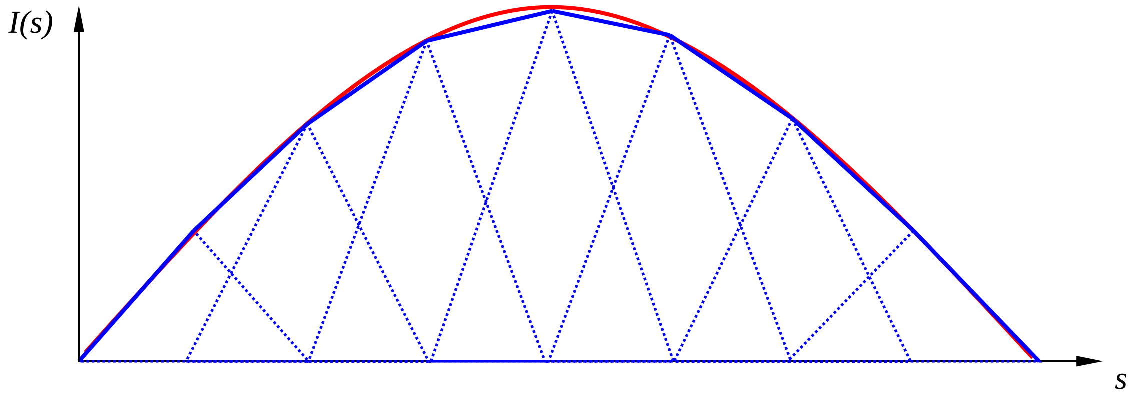


Fig. 18.2: Triangular basis functions.

This fully occupied equation system has to be solved for the unknown currents I_n. LU decomposition is used in **AN-SOF®**.

The MoM is applied by first dividing the wire structure into N segments, and then defining the basis and testing functions on the segments. Triangular basis and pulse testing functions are used in **AN-SOF®** as shown in Fig. 18.2.

When a curved wire is described parametrically by a vector function (5), the basis and testing functions are curved in the sense that their support is a curved subset of the wire. Therefore, when curved basis and testing functions are used, the Curved Method of Moments is obtained (CMoM).

In order to fill the impedance matrix [Z], an adaptive Gauss-Legendre quadrature rule is applied to compute the involved integrals.

After having solved the equation system, the currents I_n are known and other parameters of interest, such as input impedances, voltages, radiated power and directivity can be computed.

18.3 Excitation of the Structure

If a discrete voltage source is placed at the i-th segment, the corresponding element in the voltage matrix is simply equal to the voltage of the generator. Thus,

$$[U] = \begin{bmatrix} 0 \\ \cdots \\ U_i \\ \cdots \\ 0 \end{bmatrix} \tag{14}$$

When an incident plane wave is used as the excitation, each wire segment is excited by the incoming field, which has the form:

$$\mathbf{E}_i(\mathbf{r}) = \mathbf{E}_0 e^{-j\mathbf{k}\cdot\mathbf{r}} \tag{15}$$

where $\mathbf{k}$ is defined by the direction of propagation, so that $|\mathbf{k}| = k$ is the wave number, and $\mathbf{r}$ is the evaluation point, Fig. 18.3. The elements of the voltage matrix are then defined by:

$$U_m = \int_{S_m} \mathbf{E}_i(\mathbf{r}(s)) \cdot \mathbf{T}(s) ds \tag{16}$$

where the integration is performed over the m-th segment, and the vectors $\mathbf{r}(s)$ and $\mathbf{T}(s)$ are given by (5) and (6), respectively.

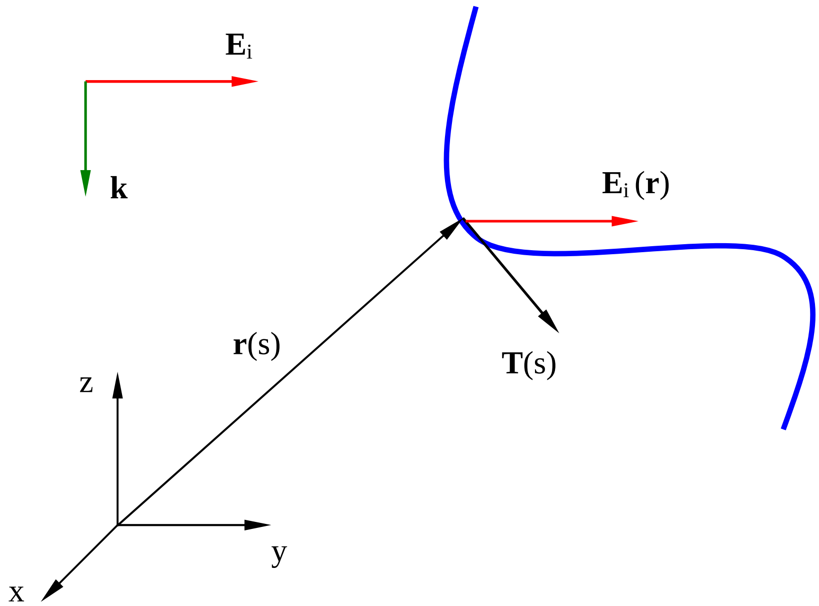


Fig. 18.3: Incident plane wave exciting a wire.

18.4 Curved vs. Straight Segments

Several examples show the advantages of using curved segments with respect to the stability and convergence properties of the solutions [9], [11]. As a consequence of the improved convergence rate, reduced computation time and memory space can be obtained for accurate results.

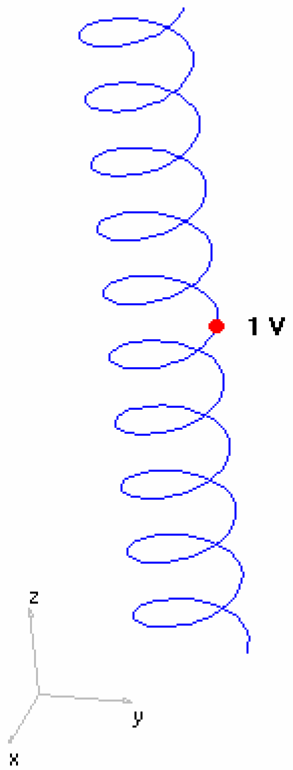


Fig. 18.4: Center-fed helical antenna (normal mode) in free space. Helix radius = 0.0273 wavelengths. Pitch = 0.0363 wavelengths. Number of turns = 10. Wire radius = 0.001 wavelengths.

As an illustration, Figs. 18.5 and 18.6 show a comparison between **AN-SOF®**, which uses curved segments, and a straight wire approximation to a normal mode helix antenna, Fig. 18.4. The convergence properties of the input impedance and admittance versus the number of unknowns are investigated.

As can be seen from these results, by using curved segments significantly fewer unknowns are needed to predict the input impedance. However, the admittance convergence is questionable for the straight wire case, while it has a notorious convergent behavior for the curved case.

The improvement depends on the geometry and frequency, but generally, if N straight segments are needed to obtain a convergent value, then N/p curved segments are needed to obtain the same value, with p = 2...10. For a straight geometry the improvement factor is p = 1, as can be expected, because there are no curved segments in this case.

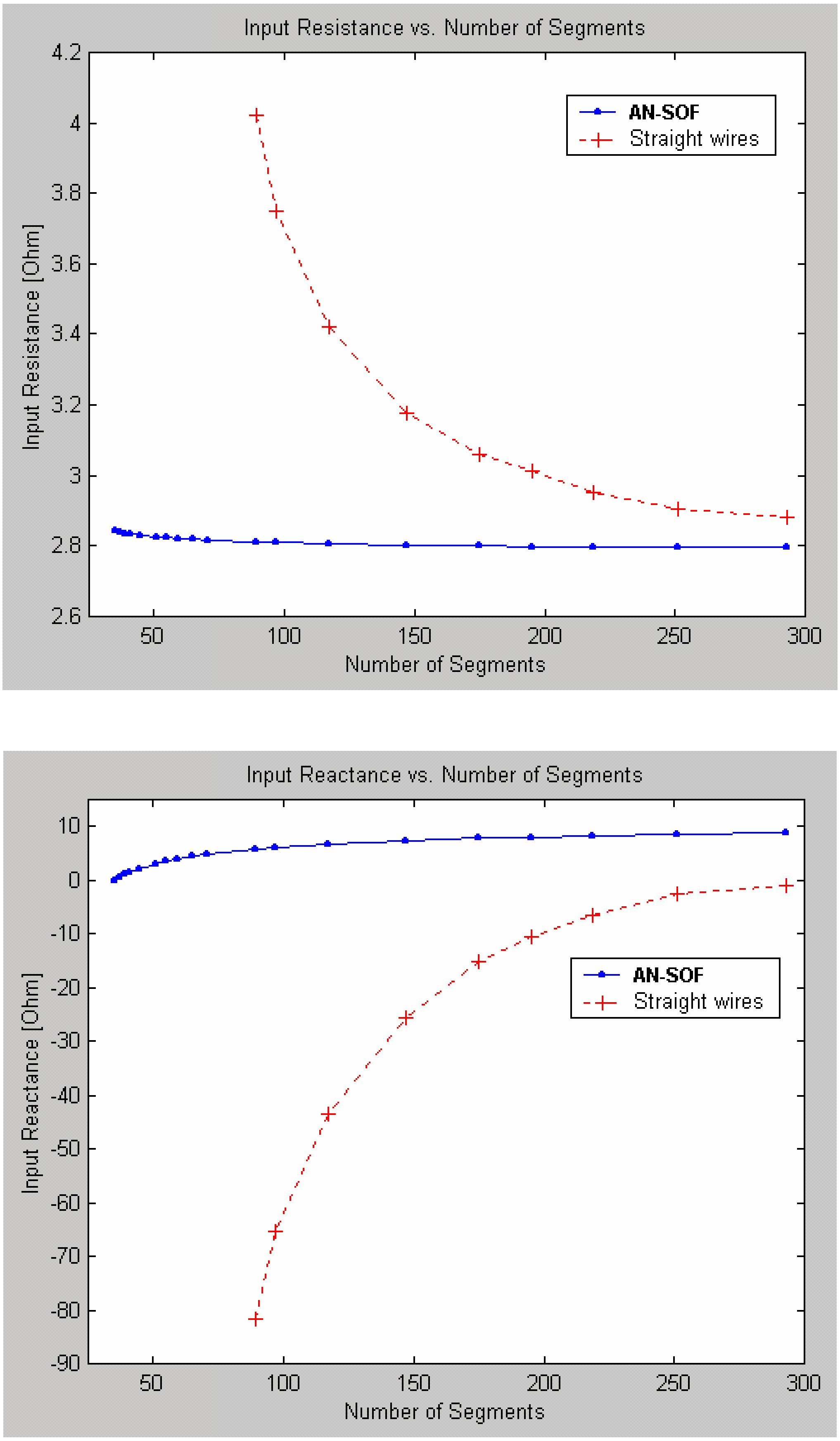


Fig. 18.5: Impedance convergence plot for the helix of Fig. 18.4.

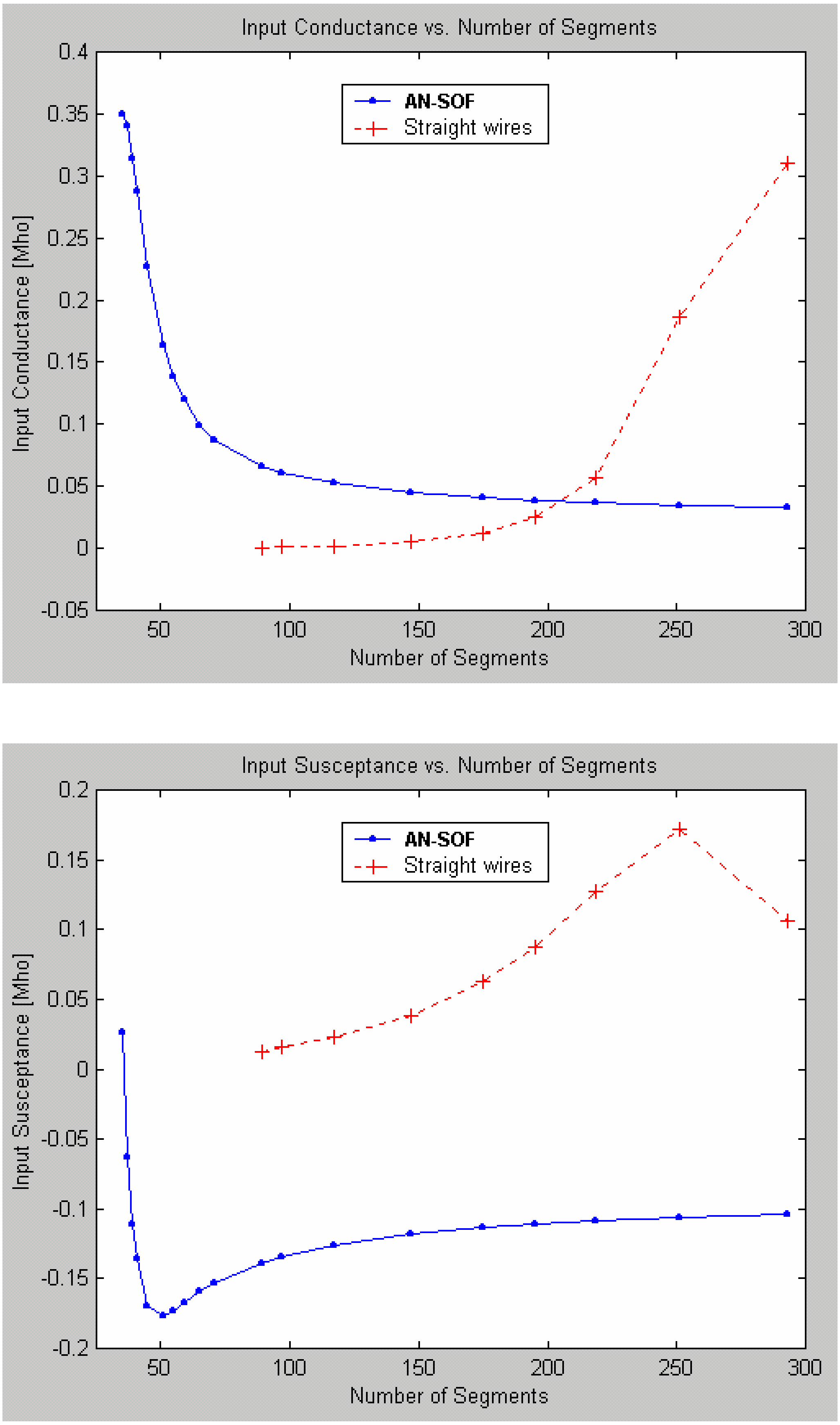


Fig. 18.6: Admittance convergence plot for the helix of Fig. 18.4.

18.5 References

[1] Harrington, R. F., *Field Computation by Moment Methods*, MacMillan, New York, 1968.

[2] N. J. Champagne II, J. T. Williams, D. R. Wilton, *"The Use of Curved Segments for Numerically Modeling Thin Wire Antennas and Scatterers,"* IEEE Trans. Antennas Propagat., vol. 40, No. 6, pp. 682-689, June 1992.

[3] Song, J. M. and Chew, W. C., *"Moment method solutions using parametric geometry"*, J. of Electromagnetic Waves and Appl., vol. 9, no. 1/2, pp. 71-83, January-February 1995.

[4] K. K. Mei, *"On the Integral Equations of Thin Wire Antennas,"* IEEE Trans. Antennas Propagat., vol. AP-13, pp. 374-378, May 1965.

[5] D. R. Wilton, C. M. Butler, *"Efficient Numerical Techniques for Solving Pocklington's Equation and Their Relationships to Other Methods,"* IEEE Trans. Antennas Propagat., (vol. AP-23, No. 5), pp. 83-86, January 1976.

[6] J. H. Richmond, *"A Wire-Grid Model for Scattering by Conducting Bodies,"* IEEE Trans. Antennas Propagat., vol. AP-14, No. 6, pp. 782-786, November 1966.

[7] R. Redlich, *"On the Extended Boundary Condition as Applied to the Dipole Antenna Problem,"* IEEE Trans. Antennas Propagat., vol. AP-32, No. 4, pp. 403-404, April 1984.

[8] D. L. Jaggard, *"On Bounding the Equivalent Radius,"* IEEE Trans. Antennas Propagat., vol. AP-28, No. 3, pp. 384-388, May 1980.

[9] G. Zhou, G. S. Smith, *"An Accurate Theoretical Model for the Thin-Wire Circular Half-Loop Antenna,"* IEEE Trans. Antennas Propagat., vol. 39, No. 8, pp. 1167-1177, August 1991.

[10] M. A. Jensen, Y. Rahmat-Samii, *"Electromagnetic Characteristics of Superquadratic Wire Loop Antennas,"* IEEE Trans. Antennas Propagat., vol. 42, No. 2, pp. 264-269, February 1994.

[11] S. K. Khamas, G. G. Cook, *"Moment-Method Analysis of Printed Wire Spirals Using Curved Piecewise Sinusoidal Subdomain Basis and Testing Functions,"* IEEE Trans. Antennas Propagat., vol. 45, No. 6, pp. 1016-1022, June 1997.

[12] S. D. Rogers, C. M. Butler, *"An Efficient Curved-Wire Integral Equation Solution Technique,"* IEEE Trans. Antennas Propagat., vol. 49, No. 1, pp. 70-79, January 2001.

Index

J

K

L

M

N

O

P

T

U

V

W

X

Y

Z